**Table of contents**

**ID= 1, Welcome to Snitz Forums 2000:**

Thank you for downloading the Snitz Forums 2000. We hope you enjoy this great tool to support your organization!

Many thanks go out to John Penfold <asp@asp-dev.com> and Tim Teal <tteal@tealnet.com> for the original source code and to all the people of Snitz Forums 2000 (http://forum.snitz.com) for continuted support of this product.

**ID = 5, Output figures**

I am an european user of Hydrus-2d. I am used to see a pF-curve of the waterconten agianst the log pressure. Also the costumers who get a report are used to see such figures. In the output of the HYdrus program it is NOT possible to create such a figure. Can you change this? I want to have the watercontent (%) theta on the x-axis and the log h (M) on the y-axis. I don't understand why such a very common figure is not included.

Hi,
Your insistance paid off. I have worked on it this morning and in the next release, there will be graphs of both h(theta) and theta(h). The same for log h. In the meantime, you can make such figures easily from the check.out file, where the table of hydraulic functions is printed.

**ID = 6, Simulation of overland flow**

I have thought of trick to simulate overland flow in hydrus-2d. I have used it very often and it works. You create an non existing toplayer of the "soiltype air". I used the sand parameters with an extreme high K value of 100 meters/day. The flow is going very fast trough this layer and under this layer I created a no flow boundary with a small slope. This forces the water to run over the no-flow boundary. I use this to simulate the waterflow over a asphaltic road into the verge with a sand soil. How this

trick can be used to simulate partly infiltrating water I don't know, maybe try a seepage face. I know this is not solid approach but it works for me. I think, in the real model a integration of the grounwaterflow and the "channel flow" is necessary to create a solid overland flow model. Unsaturated zone measuring sensor I have an idea of an sensor that is able to measure water content in the unsaturated zone in soils. It is based on the capilair capacity of holes made in an silicium plate. I first want to know If any of you know if this principle can work. Furthermore, is there anyone interested in creating such a device? The basic idea is that you create a wide range of channels with different diameters that are filled as the capillary forces are the same as present in the soil matrix. The smallest channels will be filled first and the larger will be filled later, when the soil gets wetter. When you make a wide range of different diameters you can calibrate the sensor and it is sensitive enough. (2 mu to 1000 mu) I think this is good enough.

Does anyone think this might work? And if it works why is there not such a sensor? Or is there such a device? Is there somebody able to find the time and money to create such a sensor? I know who can make such a sensor, and have more details if anyone is interested. I hope you can give me an answer.

With kind regards, Ing. IJ. H. de Haan MSc. Technical University Delft Faculty Of Civil engineering, Civil materials science group Stevinweg 1, 2628 CN Delft tel; +31 (0)15-2785710


Dear Ijsbrand,
I read your interesting comments about overland flow in the czech hydrus forum. Did you tried this with a 2D-model? Have you any further experience with overland flow in Hydrus?
I am trying to simulate a slope with H2-D, and have no idea how to integrate overland flow...
Uwe


Dear Uwe,
I used a crude way to force an overland flow in Hydrus-2d. I created a non excisting layer (air) with a very high permeability and no holding capacity. This is natturally not the way to do this. You get a strange flow patern where there are no flows present..... But It worked for me, because I had to force a flow over a non-permeable layer ( a road ). I was not interested if and how much water could infiltrate ( nothing in my case). I encountered some simular problems with a irrigation sceme on a sloping grassfield. It is not easy because the amount of water running overland is depending on the infiltration possiblities. Then you have to calculate how much water will arive in the downhill element (input border). And this is depending on how much is already infiltrated in the uphill element and the amount that is directly fallen on the element border. I believe there are some 2d models which can calculate this infiltration, but to my knowledge there is no 2d unsaturated model that does it all. (try to find the infiltration models on the internet)

It is surely better to introduce this problem to Jirka and the other who are developing the Hydrus-2d model. It is a challenge for them to be the first to incorparate this. But

maybe all the hydrologist should try to figure a way to incorpurate this and than Jirka can try to program it. Any adeas are welcome.

I hope I helped you a little bit. If you want to use my method let me know and I will tell you the details. But I have to say you have to test if it works in you situation,
IJsbrand de Haan

Hi IJsbrand de Haan
I am a Phd student at University of Western Australai, australia and I am new user to Hydrus2D. I am applying it to find out the impact of open surface drainage on water table. I want to know how I can handle the problem of overland flow as in my reserach it is very important as drains usually take infiltration excess flow as suface flow into them. I have no idea how it can be handle. Can you please send me detail of your modelling how did you do it? Thanks a lot for your help.
Mahtab Ali
PhD student
UWA, Australia
ali@cwr.uwa.edu.au

I did this as a part of my research a while ago (although I used SWMS2D, I bet you can do the same with H2D).
Assuming you have all possible surface fluxes (inwards or outwards; rootwater uptake, evap and precip), at the end of each time step either using cum-q.out or A-level.out you can do your own water balance externally against precip (if i am not wrong: h2d, removes the excess precip if inward flux is limiting).
If you have a drain, the code would simulate it's effect (nodal gravity sink) as a resultant your influx across the atm boundary would increase unless limited by a (very low Ksat) limiting layer, as against without a drain.
Cheers
Shrini

I am also working on modeling pavement water movement but i am interested in crack infiltration in addition to overland flow. Thus, instead of heaving a totally impermeable layer on the surface, i introduced some cracks which i assigned similar or slighlty less permeable properties than the air layer. For me the overland flow seems to work only when i don't have cracks. I could not get HYDRUS2d to work in presence of cracks and overland flow. I assumed the crashes were caused by high gradients and i significantly modified the mesh around the cracks to make them extremely fine but it still hasn't worked for me yet. If anyone has any ideas, i would be happy to hear about them.
Thank you,
defne

**ID = 7, Falling head infiltration boundary condition**

Hi out there,

I am trying to model falling head infiltration experiments carried out in a well with a zylindric filter section below gw. The experiment consists of adding water to the well and record the falling head with time, similar to injection slug tests. The model is radial-symmetric. All boundary conditions are clear exept the head condition in the well. Now I am wondering how to define the falling head condition in the well so that the infiltration rate through the screen interval corresponds to the rate of decrese of the water column inside the well.
Hector

Dear Eileen,
Hector needs to make a domain like below.
He puts a fictional material in the tube on the side with saturated water content of 0.99 and very high saturated hydraulic conductivity. An observation node on the bottom of the region will give him pressure verses time.
Regards,

Dear all,
I like the solution on the problem that was send in. But I have one remark does the water not infiltrate in the sides of the hole normally? I mean that not only the bottom is reacting as an infiltration layer. The way the well is designed creates the picture doesn't it?
IJ.H, de Haan

Dear Hector,
The correct solution to this problem obviously is to keep track of the water in the well (which is outside of the transport domain), integrating the infiltrating water into the soil profile, and correspondingly adjusting the water level (falling with time) in the well. This problem needs a special system-dependent boundary condition, which is not implemented at present in HYDRUS-2D. HYDRUS-2D would need to be modified to solve such problem. I will consider to implement such boundary conditions in future versions. Thanks for your suggestion.
Jirka Simunek

**ID = 8, Getting Hydrus simulation to work (challence)**

Dear all,
I am working on a simulation, that is rather complicated. The simulation represents an road with on top an impermeable layer of asphalt. The core of the road base construction is clay and that is the point of interest. The whole embankment contains 5 different soil types. 1 air, 2 clay,3 concrete granulates,4 loam and 5 sand. I use the air to simulate the water running of the road to the verge. (this worked well in other simulations). Furtehmore I have two drain pipes. And an atmospheric toplayer and the bottom is an controlled groundwatertable (pressure 0). This simulation has to run for 10 years with a daily input.

But I get an divide by zero error the whole time. I have changed almost everything to make it work. Is the simulation simply to complicated to be calculated by Hydrus?

I attached the inputfile (zipped) maybe you can all have a look en see what is wrong? I would aprecciate it very much.

Ing. IJ. H. de Haan MSc. Technical University Delft Faculty Of Civil engineering, Civil materials science group Stevinweg 1, 2628 CN Delft tel; +31 (0)15-2785710

Hello outthere,
I am experiencing right now trouble with suden infiltratio into a clay soil. Could you specify how to change the clay parameters in order to alleviate the numerical problems
Hector

IJsbrand,
I have briefly looked at your problem. I think such problem can be solved with HYDRUS (as long as you are fine with that approximation for the surface run off). However, your discretization is not adequate for that.
a) You need much finer discretization. The surface layer (which you simulate as coarse sand) needs vertical discretization on order of cm, while you have 70 cm. I would probably use for this surface approximation material less nonlinear, although more conductive.
b) You did not specify any BC for drains.
c) You use clay as one material layer. Van Genuchten-Mualem model is often not adequate for such material, because of extremely sharp decrease of conductivity close to saturation. Choose instead VG-M model with 2 cm air entry. Often much better, and certainly more stable numerically.

I do not have more time to analyze your problem since I'm leaving tomorrow for Switzerland to teach with Rien and Karim HYDRUS course in Zurich. But I hope that you find my comments useful.

Jirka

## ID = 9, Atmospheric boundary conditions

Dear HYDRUS Users,
I have questions regarding simulations using atmospheric boundary conditions.
1.How do I determine infiltration and actual evaporation rates from HYDRUS outputs? Because HYDRUS uses both precipitation and potentail evaporation as the limiting fluxes, how could I know from the actual surface flux that the maximum values have been reached?
2.Should infiltration be taken from the actual surface flux of the first iteration and actual evaporation be taken from the flux of the last iteration?

3.Should the net inflow in mass balance equal the net amount of (actual surface flux+actual transpiration+drainage)? If so, why does the reported net inflow not match with change in water storage obtained from the water content profile?

Preecha

Dear Preecha,
1+2) When you specify at the same time infiltration (precipitation) and evaporation at the same boundary (atmospheric BC), HYDRUS will calculate the potential flux across the boundary as flux=prec-evap, and then applies this resulting flux. You can see based on the sign whether the flux is into (infiltration) or out of (evaporation) the transport domain. HYDRUS prints into the output files only this potential flux (not precipitation and evaporation fluxes), as well as the actual flux. When you compare the potential and actual fluxes, you can immediately see whether the potential flux was reduced due to the soil conditions.

3) The net inflow should be equal to the change in storage. You can see the mass balance error in the "mass balance information" output. When you use appropriate spatial discretization, this error should be smaller than 1%.

Jirka Simunek

**ID = 10, Absolute error in the water mass balance**

Dear HYDRUS Users,
I try to verify the values for the absolute error in the water mass balance reported in "balance.out" file. >From equation 5.24 in the manual, the absolute error is equal to the difference in water volumes at time t and at initial time+cumulative transpiration-cumulative surface flux+cumulative drainage. However, I cannot match the values reported for the absolute error in the water mass balance. Does anyone have any suggestions on this matter?

Regards, Preecha

Dear Preecha,
Do you mean you recalculted the mass balance error and this is not the same as reported in the outputfile? In general, it has to be all water initial present minus all outward flows + all inward flows is the total amount of water present. The deviation from that amount is the mass balance error.

IJsbrand de Haan.

Preecha,
I do not see any reason why the mass balance reported in the "balance.out" file should not correspond with the other outputs, mainly cumulative boundary fluxes, as long as the numerical solution converged. Just add all the cumulative boundary fluxes,

including root water uptake, from the Cum_q.out file and compare that with the volume of water in the transport domain at the beginning and end of the simulation. That's exactly the number reported as mass balance error.

Jirka Simunek

I apologize for my mistake to ask question about Hydrus 1D in the Hydrus 2D discussion forum.
And I want to say thank you for your prompt reply. And you are right, the area I'm studying is a very dry area and the value I have for transpiration and evaporation are very high.
I have another query I tried to insert all the data I'm using in Hydrus 1D in Hydrus 2D, considering a column (150cm*100cm). The mass balance error I have in Hydrus 1d is very low (<< 1) but using the same value for the 2D simulation the mass balance error is very high (also 22.55). Why this happen?
And also I'm studying a landfill cover, so below my cover (my column) there is garbage, I was wondering if is correct to use "free drainage" like bottom BC.
Is it possible estimate water flow parameter for garbage? Can you suggest me any papers about this topic?

Thank you in advance,
Fru

The numerical solution in HYDRUS-2D is very similar (except for dimensions) as in HYDRUS-1D and thus the mass balance error should be similar. HYDRUS-2D typically has the mass balance errors on the same order as HYDRUS-1D. You must have something very different, e.g., discretization, soil hydraulic properties, or boundary conditions.
Free drainage seems OK. Only in case that the garbage is very course, i.e., water does not enter into garbage until the bottom of the cover is saturated, then it may be better to use seepage face.
Soil hydraulic properties for garbage: I do not know. I believe that there was some discussion on that already here in this discussion forum.

Jirka

## ID = 11, Assignment of properties

Hello NG,
we know, that in HYDRUS - unlike e.g. groundwater models -the evaluation of the material properties is handled nodewise. Depending on the spatial discretization this will lead to a "smearing" of the textural contrast expected at material intefaces. Capillary barrier effects for example are somewhat unserestimated. To minimize these effects requires an excessive mesh refinement.

Is there anybody out there, who has rewritten the source ov Version 2, so that the properties are evaluated at the element ?
Hector


Traditionally the soil hydraulic and transport properties are assigned to nodes and they change between the nodes depending on the interpolation scheme (linearly for linear finite elements) (see discussion in van Genuchten [1978]). The nodal value of the independent variable (typically the pressure head, h) is used to obtain the nodal values of the dependent variables (the water content, q, the hydraulic conductivity, K, and the hydraulic capacity, C). When the soil hydraulic properties are assigned to nodes then only one value for each dependent variable is obtained at each node. The material properties are thus smoothed between the neighboring nodes and the solution for water flow and/or solute transport is obtained for this smooth material interface (see Fig. below). Although such solution is adequate for most natural problems where the soil material boundaries are not well defined and are typically smooth, the solution is inadequate for such engineering structures, such as capillary barriers, that depend on the abrupt material boundaries. The numerical solution in the HYDRUS-2D code need to be modified (and I remember that I have done that for you in version 1.0) to allow for abrupt material boundaries as follows. The material properties need to be assigned, contrary to the traditional solution, to elements. The independent variable, the pressure head h, will still be smooth and continuous. The dependent variables (the water content, q, the hydraulic conductivity, K, and the hydraulic capacity, C) are then evaluated from the nodal value of pressure head for each element independently. In one-dimensional problem, one can have two different values of dependent variable if these are evaluated for the upper (qi-, Ki-, and Ci-) or lower (qi+, Ki+, and CI+) element (see Fig. below). For two-dimensional, and higher-order, problems, one can have as many values of dependent variable in one node as elements with different material properties. Thus, the dependent variable can change abruptly in a node, when the material properties are changed.

Jirka.


Jirka,
thank you for the advises. I completely agree with your statement, that for "natural problems" the handling of node wise assignment of properties is adequate. In the postprocessing this approach is considerably more straight forward, due to the smooth transition of the dependent variables. I have not had a look at the code yet, but considering the many features (hysteresis, temperature etc.) I guess things are pretty comfortable doing the node wise assignment. The element approach would have to run loops through different material domains, considering different dependent variables at the edges. That different structure might impact the handling of the different modules integrated in the code.

I often have to do with geotechnical structures with more or less defined material boundaries. You kindly adapted the Version 1 code to assign properties to elements. Herein one has to define both materials and regions ("layers") to the pertinent property. A bit more work, but things worked fine. Now I have upgraded to Version 2 and the purpose of my request was to identify others who may as well have the

requirement for abrupt material changes at elemental boundaries and see how we could proceed.

Hi,
I have implemented an option in the 2.x version of HYDRUS2 code that allows users to assing properties into elements instead of nodes. Interested registered users of the HYDRUS-2D software package can contact me and I will send them an alternative h2D_calc.exe code that does that. This alternative version of h2d_calc.exe uses codes for sublayers as material assignement.
**ID = 12, Soil concentrations**

Dear colleagues
In our experiments, we often take soil samples from some layers (e.g. between 0-10 cm, 10-20 cm, 20-30 cm....) and soil concentrations analyzed are subject to these layers. However, observations in Hydrus2D are subject to points. Even I could convert soil concentration from the water concentration, I still have great difficulty in comparing point-data to the layer-data.

Another way to do it is to covert the results to .txt files for water contents and concentrations for all the print out times for the whole model domain, then integrate the water contents and concentrations for these layers, and then calculate the soil concentrations. But this is very troublesome.

Best regards
Liping

Liping,
It is quite overwhelming in how many different ways one can define concentration in the objective function for parameter optimization. The following options come quickly in mind:
a) resident concentration b) flux concentration c) resident concentration in the equilibrium (mobile) phase d) resident concentration in the nonequilibrium (immobile/kinetically sorbed) phase e) total liquid concentration (mobile+immobile) - something measured by TDR. f) total solute (liquid + solid). How do you define concentration here? (theta*c+ro*s)/theta?

Some of these can be area-averaged (I guess all except flux concentration).
Are you personally interested in the total solute (liquid + adsorbed) or only liquid phase concentrations?
Jirka

Dear Jirka
I think it is very necessary to implement the area water contents and soil concentrations for particular layers into the Hydrus2D code. In our field pesticide leaching trails, we monitor soil water concentrations, soil concentrations and contents.

Because the limitation of the objective function in current Hydrus2D code, parameterising pesticide transport could be based on only the soil water concentrations. We are trying to compare results from Hydrus2D with those from GLEAMS and LEACHM, but there is a blank in soil concentrations for Hydrus2D.

I do wish this function could be incorporated into the code as early as possible so that I could make use of more our observed data with Hydrus2D.
Liping

Liping,
I have implemented this for the area evaraged water contents for particular layers in HYDRUS-1D, but not yet for concentrations, and not yet in HYDRUS-2D. Obviously the best solution would be to implement it directly into the code itself. Let me know how much you really need it and I will put it on my "to-do" list.
Jirka Simunek

## ID = 13, Upflux through free drainage lower boundary

Dear Jirka,
I am simulating a root zone water balance with hourly to daily values of evapotranspiration and precipitation over the course of three years. I have an atmospheric upper boundary and a free drainage lower boundary. The simulation completes nicely with mass balance errors less than 0.1% for each approximately 30 day period. However, the cumulative bottom flux is showing a positive flux in year 2 (on the order of 2 mm). By year 3, the cumulative bottom flux changes to being negative and proceeds as would be expected with flow downward out of the model domain. I am wondering why this is happening if the free drainage boundary fixes a zero gradient condition at the bottom of the domain. Could it be that the 2mm is still within the model error? My total profile depth is 122cm and Qs is 0.39. Thanks in advance for your help.

Best Regards,
Jason Smesrud
CH2M HILL 825 NE Multnomah, Suite 1300 Portland, OR 97232-2146

Jason,
Free drainage lower boundary condition represents a gravitational flow at the bottom of the soil profile. As such it can not result in positive (upward) flux. It must always be negative, i.e. downward. I guess you must have had some periods when the code did not converge. That could result in positive bottom flux. Obviously, 2 mm flux in a year is a very small value and may have been within the precision tolerance. You may try to use stricter water content tolerances.

Jirka Simunek

**ID = 14, Values for R2H and R2L**

Dear HYDRUS Users,
I have a question regarding to the input values for the maximum potential transpiration rates (R2H and R2L) in water stress parameters. Should not these values be constant but corresponding to the daily potential transpiration rates assigned in the table of time variable of boundary conditions? Could anyone give me a suggestion of what exactly the values to be assigned for these 2 parameters?

Preecha

Preecha,
Yes, we have misunderstood each other.
One can specify as many potential evaporation rates as you wish in the "Time-Variable Boundary Conditions" dialog windows. The values r2H and r2L do not have anything to do with that. As long as the soil is wet and the pressure head is larger then h2, the actual uptake will be the same as potential uptake. Only when the soil becomes dry, the potential uptake will be reduced to the actual uptake. When this reduction starts then depends on these two values - r2H and r2L, because they define the pressure head h2 when reduction starts. Apparently, some people believe that when this reduction starts depend on the potential transpiration rate; if the potential transpiration is larger the reduction starts earlier. If you do not agree with that then you can specify r2H= r2L and h2L=h2H.

Jirka Simunek

**ID = 15, Flexibility with more than one variable head bound**

After a brief insight into the code of version 2.0 I found an elegant modification in respect to the time dependent pressure head condition. Now nodes on such an variable head boundary can be at any elevation whereas in the prior version nodes assigned this boundary had to be at the same level. This offers much flexibility in dealing with sloped boundaries, as I often do. This changes were however not reported in the documentation (or did I oversee it ?)

Flexibility could be considerably increased, if more than one time dependent boundary conditions could be as-signed. Imagine a dam bounded at each side by different water levels changing independently in time. If one could assign KODE(n) any number (lets say from 30 - 39) in the Boundary Conditions module and would then define these time series in atmos.in (or open a new file containing only time variable head information) then the desired flexibility would be gained.

Hi,
In the version 2.x of hydrus-2d, the time-dependent pressure head boundary is assigned to the node with the lowest vertical coordinate (of the selected boundary) and the pressure head in other nodes is adjusted dependent on the vertical coordinate (hydrostatic equilibrium from the lowest located node).

I agree that it would be usefull to have multiple time-variable pressure head boundaries. I will try to implement it into the next version of HYDRUS.

## ID = 16, Hydraulic potential output

Hello Jirka & devolopers
now that we are compiling recomendations/wishes for the next "Hydrus-Santa-Claus-Presents-List" here goes another one:
Dealing in geotechnics / hydraulic engineering the spatial distribution of hydraulic potential (pressure head + elevation) yields a more concise insight into the flow process than the pressure field does. The output-routine could be modified to consider an binary file (e.g. hpot.out) for hydraulic head. This file could be generated additionally to the h.out file. Then it has to be renamed (e.g. th.out) and can then be viewed in the post-processing module.
Alternatively, a logical variable (lhpot) passed over in e.g. selector.in could control if pressure or hydraulic head is output.

Hector

Dear Hector,
I see that we are proceeding in the right direction. I got similar suggestion, i.e., to print in addition to pressure heads also total heads, earlier this year and implemented it in quite similar way as you suggested. I have a version of the HYDRUS2 code that prints out a new output file hT.out (h.out is for the pressure heads). Once you rename it (to h.out) you can display total heads instead of pressure heads in the Graphics module.

Jirka

## ID = 17, Variable boundary head

Greetings.
When I have a river and the river stage is fluctuated from $z=4$ m to $z=5.5$ m and the river bottom is at $z=1.0$ m and I need to use this river boundary as a variable boundary head, did the variable boundary head should extend from $z=1.0$ to $z=5.5$ or from $z=1$ to $z=4$ and when the river stage is going down after rising how I can present the seepage face.

Mo'ayyad

Dear Mo'ayyad,
You should specify the variable boundary head from $z=1.0$ to $z=5.5$. However, when the water level drops, the code will prescribe fixed negative pressure above the water level. You need to split the problem into two. First run one BC (5.5m), then copy the input, import result, and restart with a new BC and seepage face specified between 4 and 5.5.

Current version of HYDRUS-2D (2.0x) allows users to specify only one type of boundary condition at any particular part of the boundary. Thus one can specify Atmospheric BC on one part, variable head on another, and free drainage yet on another part. But these types of boundary conditions are fixed in time.

Many actual situations are, however, more complex, and even the type of the BC (not just the value, such as precipitation) must change dynamically. A typical example is a variable head BC (i.e., for a stream and/or river). When the water level in the stream drops, part of the boundary may become a seepage face, atmospheric boundary, or no flow boundary. Another typical example is a non-active seepage face, where one can describe atmospheric BC (it rains on seepage face). We are currently working on some of these dynamic combinations (as given above for variable pressure head or seepage face). We would welcome your suggestions what other types of BC's and their dynamic combinations you could use. Thanks.

Jirka Simunek

### ID = 18, Time stepping schemes for moisture transport

Respected Sir,
I have few questions related moisture transport studies under a point source emitter. While simulating a system from t=0 to t=first time step the initial conditions were assumed constant over the whole domain. That lead to a solution domain inconsistent to the physical theory i.e. the trend of moisture distribution is not according to the desired trend and the same trend just get carried over the successive time step. Now I have a doubt .. While assuming a initial condition, the physical parameters for the next time step were assumed for half time-step. Which itself is constant over the whole domain thus not conforming to the desird trend. Should I use an assumed trend for the half time-step to ascertain the parameters to be used for the next timestep or any thing else. I have tried hard with that option, but failed.

It would be so kind of you if u can suggest me the way I can come over this problem. For your information I am a Ph.D. Scholar at Dept. of Agril & Food Engg. I.I.T., Kharagpur and working on FE modeling under a point source emitter. I am trying to write my source code in C language and a significant time ahead of my Ph.D.

D.R.Sena Ph.D. Scholar Dept. of Agril. & Food Engg. I.I.T., Kharagpur - 721 302 (West Bengal) INDIA e-mail: drsena@agfe.iitkgp.ernet.in

### ID = 19, Detection of critical gradients in a domain

Hello out there,
when inspecting geotechnical problems, sometimes the detection of critical hydraulic gradients at material interfaces is of interest. If not designed under consideration of filter stability criteria, hydraulic gradients may lead to material transport.

The gradients cannot be deduced from the flowfield, since the later is represented by the product of gradient times conductivity.

Below you can find my suggestion to compute the directional compenents of the gradient field. However, I am not yet sure how the smearig effect of nodal properties assignment is to be considered. Any hints appreciated !

Similar to the output of hydraulic head, the output may be controlled by a logical variable (lgradout?)in Version 2.x

Hector

Calculate Gradients i each coordinate direction

```
subroutine          GradHOut(Gradhx,Gradhz,x,y,t,ierr,NumNP,NumEl,          !
NumElD,hNew,KX,ListNE)           integer          e          dimension
hNew(NumNP),x(NumNP),y(NumNP),ListNE(NumNP),    !    KX(NumElD,4),    !
List(3),hpot(3), ! Gradhx(NumNP),Gradhz(NumNP)

do 11 i=1,NumNP Gradhx(i)=0. Gradhz(i)=0. 11 continue do 14 e=1,NumEl
NCorn=4 if(KX(e,3).eq.KX(e,4)) NCorn=3 do 13 n=1,NCorn-2 i=KX(e,1)
j=KX(e,n+1) k=KX(e,n+2) List(1)=i List(2)=j List(3)=k c hydraulic Potential
hpot(1)=hnew(i)+y(i) hpot(2)=hnew(j)+y(j) hpot(3)=hnew(k)+y(k) c FEI-Matrix
vi=y(j)-y(k) vj=y(k)-y(i) vk=y(i)-y(j) wi=x(k)-x(j) wj=x(i)-x(k) wk=x(j)-x(i)
Area=.5*(wk*vj-wj*vk) A=1./Area/2.

c          hydraulic          Gradient          H.          Montenegro          Dec.          2000
Gradhxx=A*(vi*hpot(1)+vj*hpot(2)+vk*hpot(3))
Gradhzz=A*(wi*hpot(1)+wj*hpot(2)+wk*hpot(3))

do 12 m=1,3 l=List(m) Gradhx(l)=Gradhx(l)-Gradhxx Gradhz(l)=Gradhz(l)-Gradhzz
12 continue 13 continue 14 continue c arithmetical Averaging (erroneous on material
boundaries???) c ListNE(i) number of elements with the same nodal point do 15
i=1,NumNP Gradhx(i)=Gradhx(i)/ListNE(i) Gradhz(i)=Gradhz(i)/ListNE(i) 15
continue

write(85,err=901) t write(85,err=901) (Gradhx(j),j=1,NumNP)

write(86,err=901) t write(86,err=901) (Gradhz(j),j=1,NumNP) return

901 print *, 'Error when writing gradients into output file' ierr=1

return end
```

Hector,
Thanks for another suggestion.
I do not think that the way material properties are assigned (either to elements or nodes) has any effect on how the gradients are calculated. It has effect only on how

the velocities are calculated since then one has to consider conductivities (and they depend on soil hydraulic properties). The gradients themselves do not depend on that.

Jirka


## ID = 20, Hydraulic total head

I have done some simulations with Hydrus-2d for a third party. They don't have Hydrus-2d and do not intend to buy it. Thats why they let me do the simulations, he, he,. But now I wan't to present the results to them and the best way to do so is with hydrus-2d,I think. Now was my (smart?) idea to use the (shareware?) demo) version of hydrus to let them run my result files. But it seems that it is not possible to view my files in this version, is this correct? And if so, is it difficult to make a version that can demonstrate results made with hydrus but without being the whole program? Maybe disable saving, printing, running of the program and altering the files? I think that it is maybe possible to show simulations done but I don't know. I am waiting on your answer.

IJsbrand de Haan


Dear IJsbrand,
There are no restrictions on displaying results of existing HYDRUS-2d projects with a demo version. Similarly as you can see all the text examples (installed with the HYDRUS-2D code) , your colleagues should be able to view the projects created by you. You just need to send them the project folder and the project.h2d file.

Jirka


## ID = 21, Number of subregions

Hi,
Is it possible to get more than 10 subregions in one run? It´s a long slope I want to see...
Another question: can anyone explain the parameters in the H2D for root distribution - vertical: lambda and A - horizontal: delta can´t find anything in the helpfiles/manual
regards
Uwe


Uwe,
a) The HYDRUS-2D allows only 10 subregions in one simulation run.
b) Root distribution parameters. This command (Options->Parameters for Root Distribution) in the Boundary module was not part of the version 2.0 of HYDRUS-2D. I added it only last year when I was working with Jasper Vrugt and Jan Hopmans on the spatial distribution of roots (and therefore it is not described in the manual, which will be updated only for version 3.0). Below are two references where you can find all details about this root distribution model.

Vrugt, J. A., J. W. Hopmans, and J. Simunek, Calibration of a two-dimensional root water uptake model for a sprinkler-irrigated almond tree, Soil Sci. Soc. Am. J., 65(4), 1027-1037, 2001.

Vrugt, J.A., M. T. van Wijk, J. W. Hopmans, and J. Simunek, One-, two-, and three-dimensional root water uptake functions for transient modeling, Water Resour. Res., 37(10), 2457-2470, 2001.

Jirka Simunek

## ID = 22, Mass transfer coef alpha in physical nonequlibrium

Hej
I try to model a solute transport using the physical nonequilibrium. For that I have to figure out the mass transfer coefficient alpha between the mobil and immobil part of the water content. Now my question: In which way the mass transfer coefficient alpha is arranged in the advection dispersion equation for solute transport.

Markus Einecke

Markus,
You can see best how the mass transfer coefficient is implemented into the mobile-immobile physical nonequilibrium model directly from the governing equations. This would be equation 3.19 (numerical implementation is discussed in Section 6.3) in the HYDRUS-1D manual and equation 3.18 (and numerics again in section 6.3) in the HYDRUS-2D manual.

Jirka

## ID = 23, Inverse solution

Hi there
I'm student and for my thesis we did (among other things) a sprinkling experiment. we installed buriable tdr waveguides in a 5*10*1 m soil block in a 3*4*4 grid (3 dimensions). during the experment they measured every 7 minutes and then over 3 months (with rainfall) every hour. additionally we measured the outflow in three soil layers. now i am about to analyse and display the results. i read a little bit about hydrus (and soil physiscs - i am in forestry). so what i wanted to know if there is a possibility to analyse and diplay the results in hydrus 2d (reduce the probblem to a vertical plane) with an inverse solution of the richards equ. i have the soil water characteristic. i know that there are many problems (water content, macropores...). but still it would help me if i would know in which direction to go. thanks for your reply and greetings from austria
Jost Georg

George,
This is exactly what the program was written for: to analyze water flow in soils. HYDRUS can solve both direct as well as inverse (i.e. calibration) problem.

Jirka Simunek


## ID = 24, Trouble aligning edges on polylines

Hello out there,
I was trying to create a vertical plane computational domain with triangular FE. I wanted to define a flat region crossing the domain in a diagonal fashion. (Kind of a tilted layer). It was possible to define the polylines. However the FE-meshing failed: The mesh edges were alined to the polylines defining the layer only with the fundamental option. The general option yielded nodes at the points of the polyline, but no edges, which is incceptable for the definition of different regions.

Interesting, this kind of problems did not appear in the previos hydrus/meshgen version.
Did anybody experience similar troubles? Am I doing something wrong?

Hector


Dear Hector,
this problem is caused by an error in Meshgen. I'll fix it as soon as possible and then I'll send you fixed DLLs.
Now, if you are in a hurry, you can try to use a little bit "tricky" solution: try to add two internal lines (lets call them P1 and P2) - one above and one under your existing diagonal line (lets call it D). These lines P1, P2 should be parallel to D and their distance from D should be about average FE mesh size on D. I've tried this and it worked but unfortunately there is no guarantee that this trick will give you always the correct mesh.

Mirek


Thank you for your assitance Mirek. I was starting to think the mistake was on my side.... I did not try out you trick, instead I used the older Meshgen version, generetad V1.1 Hydrus files and converted them to V.2.0 and run them. There some problemes arouse: I converted the boundary condtions file to ASCII, and could change stuff (angle of anisotropy etc.) but then Hydrus would not run. So I could check all my imputs were correct at the preprocessing level but the computation module would not read input correctly. Fortunatly things worked, when I converted ASCII back to binary input. Hydrus2D could handle that.
Hector


Dear Hector:

You can download a fixed version at our new page "Hydrus-2D Downloads".
Mirek


**ID = 25, Assignement of Anisotropy ratio & angle**

Hi outthere,
working with fracture media recently I tried the anisotropy option. I do not know if the definition of the angle is documented and so I had to find out: positive in clockwise direction. In other GW-programmes I worked with, it was always in counter clockwise direction.

When defining anisotropy one has to prescribe the ratio of the components and the angle of the main components with the local coordinate system. Alltogether 3 Values have to be defined. Dealing with complex structures this can be quite thorny: you have to select 3 times ele-ments and assign values. One way to handle this is converting into ASCII and shifting the values via a serch & change function.

My question: is it possible to assign the 3 necessary values for definition of anisotropy in the pre-processing menu?
Regards
Hector


Hector,
As usual you have exactly pin-pointed where we could improve the graphical interface. At this moment one needs to make a domain selection and then specify each anisotrophy parameter independently. If I understand correctly you suggest that we should allow users an option to select transport domain once and then assign all three anisotrophy parameters at once. That's a good idea and we will do that. Thanks.

Jirka


**ID = 26, Isolineplot velocity distribution**

I am wondering weather the Isolineplot of velocity distribution displays the value of one directional component or the absolute value $v=sqrt(vx**2+vz**2)$ of darcian fluxes at the nodes?
Could not find in the reference manual, so any hints are appreciated.
Hector


Hello Hector,
The absolute value of velocity ($v=sqrt(vx**2+vz**2)$) is displayed.
Regards Mirek

**ID = 27, Snowmelt**

I am just beinning to use the Hydrus 2d model to look at infiltration of a heapleach field cap in Northern Nevada. I was wondering if anyone has looked at ppt. in snow form and how people have addressed applying this to the model. I was going to use "atmospheric boundry conditions" at the soil surface but know that the majority of our spring surface water available for infiltration is from snowmelt. How has this been dealt with by other users???

Thanks Catherine Davis cadavis@ltol.com

Catherine,
Here are some comments to the snowmelt issues. Ideally a coupling of the process based snowmelt model is required with HYDRUS. The mass/energy balance of the snowpack can use atmospheric boundary conditions at the top and a specified head/temperature, gradient or flux BC at the snow soil interface. If the snowpack is well established, then the temperature at the interface is fairly constant (0 to minus 0.5 deg C).

In 1996, I wrote such a model (see references below) to get meltwater flux rates. These were then taken as input into HYDRUS to predict flux through the soil. In a way, a decoupled approach may be better because one would avoid problems in numerical oscillations resulting from freezing and thawing of snowpack and the resulting morphological changes in the snowpack (narrow range of liquid water content between dry and wet snow).

You can use any physically based snowmelt model (eg. SNTHERM by Rachel Jordan from US CRREL, DAISY/IHDM by E.M. Morris or UCGVDSM etc.). My feeling is that degree day approach is not a good one to simulate dry and wet snow conditions. Most degree day models use energy transport for dry snow conditions and mass transport for wet snowpack conditions. They are usually not good for transitional conditions.

Regards Narendra

Catherine,
HYDRUS codes at this moment do not deal with snow-melt issues at all. You need to specify fluxes at the soil surface that can actually infiltrate into the soil profile. Snow will not do. You can use something as the degree day concept and calculate in advance when snow accumulates at the soil surface and whet it will actually infiltrate into the soil profile. I have recently implemented it into HYDRUS-1D, but not yet into HYDRUS-2D.
But there is still hope. In March I will have a visiting PhD student who plans to do just that - to implement into HYDRUS-2D the thaw-melt routines.

Best Jirka

**ID = 28, Variable head boundaries REVISITED**

Hi all,
In the version 2.x of hydrus-2d, the time-dependent pressure head boundary is assigned to the node with the lowest vertical coordinate (of the selected boundary) and the pressure head in other nodes is adjusted dependent on the vertical coordinate (hydrostatic equilibrium from the lowest located node.

However, the prescription of a hydrostatic pressure distribution can lead to serious convergence Problems, when the expected pressure change GW or river stage rise)are large. Consider that the rise of waterlevel at the boundary of a dam can take several meters. The dam itself canbe quite wet t the beginning of the calculation, then The variable head boundary would be several m negative pressure at the top nodes of the boundary.

To my opinion,it would be better in these cases to assign hydrostatic pressure below the prescribed waterlevel (ht in atmosph.in), while taking the initial pressure above the water level or some capillary fringe. This assigment would diminish large gradients at the start of computation.

Is there any physically based argument for assigning hydrostatic pressure above the seepage face?

Hector


Dear Hector,
The natural boundary conditions are obviously much more complex and dynamic than those in version 2.0 of HYDRUS-2D. I have been struggling last year in developing more complex boundary conditions for the next version. The dynamic interface between the stream (variable head BC) and dry slope attached to water level (seepage face) is one of the options I have been working on. Another is that in version 2.0 atmospheric BC could not be applied on the seepage face (two BC at one location), which seems logical - it rains on nonactive seepage face. Below is a brief list of various new options that I'm developing for HYDRUS.

Dynamic boundary conditions - switching at given time between head and flux - switching between seepage face and atmospheric BC based on the system's status (streams) - switching between given pressure head, atmospheric BC, and seepage face based on the system's status (streams) - multiple time-dependent boundary conditions - wells

Jirka Simunek


Hi all,
Thinking about the implementation of time variable GW-level changes, it becomes clearer that only a condition change from time variable (kode=3) to seepageface

(kode=+-2) depending on the boundary node's location with respect to the GW-level would lead to a physically based flow regime on slopes. Assignement of hydrostatic pressure leads to unreasonable pressure gradients above the capillary fringe, especially during rapidly falling GW/river stage-levels.
Any idea, if this kind of condition type shifting is feasible?
Hector


Yes, I'm implementing this dynamic change in BC.
J.


## ID = 29, Units

Hi All, I have recently started using HYDRUS2D. Currently, I am trying to model spread of an Arsenic plume but I think i have a problem with the units.

I opted to use grams, meters, and days for units.I input all my parameters such as bulk density, initial concentration, lateral and transver dispersivity in units of grams, meters, and days and when i get the concentration output i assume the units for it are also g/m3.Would you let me know if this approach is wrong. If one has units of mmol for mass then, how does this translate for inputing a value for bulk density? Shouldn't one input everything in consistent units?

I am also assuming (based on linear adsorption simulations that show the spread of the plume retarded almost at the same rate as the retardation coefficient) that HYDRUS2D outputs total contaminant concentration, not the aqueous or the adsorbed phase. If this is so, do you have any ideas why my nonlinear adsorption runs show that the mass dissapears with time? My linear adsorption runs do show the mass moving and being retarded, whereas my nonlinear adsorption simulations just show the mass dissapearing with time.

Defne Apul


Defne,
One needs to use the same units throughout the input, i.e. time and lenght units must be everywhere the same. Also the concentration units should preferable use the same units as when designing transport domain, etc. But for concentration units it is necessary to keep this convention. Since concentration term appear in every term of the CD equation concentration units can be independent of spacial units. This means that one can use units such as mg/L even when length units are in meters. Another spatial case is the bulk density and Kd. Since there two variables are in CD equation as product, their units can also be independent from units of other variables, providing they are the same (for Kd and ro). One can figure out most of this just by analyzing convection-dispersion equation.

Regards, Jirka

**ID = 30, Variable head boundaries / field capacity concept**

Hello out there, I made some modifications in subroutine SetAtm to the way, time variable head is implemented. These consist of limiting the hydrostatic pressure distribution above GW-level to one meter. Above that pressure is kept constant to - 1 m, if pressure at the previous time level was less than - 1 m.

This modifications eliminate assignement of sharp gradients at high levels above GW. At the same time it seems to me to be more consistent with the field capacity concept. The value of 1 m was chosen arbitrarily and things worked well. However this value could be passed over as variable in e.g. atmosph.in.

Depending on the ratio of the materials conductivity and the velocity of water-level fall, the consistent boundary condition would be the shifting from time varying to seepage face.

Hector

Dear Hector,
The natural boundary conditions are obviously much more complex and dynamic than those in version 2.0 of HYDRUS-2D. I have been struggling last year in developing more complex boundary conditions for the next version. The dynamic interface between the stream (variable head BC) and dry slope attached to water level (seepage face) is one of the options I have been working on. Another is that in version 2.0 atmospheric BC could not be applied on the seepage face (two BC at one location), which seems logical - it rains on nonactive seepage face. Below is a brief list of various new options that I'm developing for HYDRUS.

Dynamic boundary conditions - switching at given time between head and flux - switching between seepage face and atmospheric BC based on the system's status (streams) - switching between given pressure head, atmospheric BC, and seepage face based on the system's status (streams) - multiple time-dependent boundary conditions - wells

Jirka Simunek

**ID = 31, Mass balance error**

Dear HYDRUS users,
I am simulating consecutive runs(days) by importing the last conditions to be the initial conditions of the next run. For the first run, I had adjusted spatial discretization such that the daily mass balance errors were within 1%. However, for the next consecutive runs, the mass balance errors became much higher at some points in time. To reduce the error down, I could only adjust time discretization since the space discretization has to kept the same. But, the errors were still high. What can I do to solve this problem?

Since hydrus reports the mass balance error based on the cumulative fluxes and does not report change in storage, to obtain the daily mass balance error, can I just simply subtract the reported absolute errors of the consecutive days?

Preecha

Preecha,
Since HYDRUS uses mass conservative solution of the Richards equation, the mass balances are in general very small. If you obtain large mass balance errors, it should be a flag that your discretization is not sufficiently fine and you should start from the beginning with finer discretization.

Hydrus report for every print time the mass of water and solutes stored in the entire transport domain and specified subregions. It also print at these time the actual change of mass for these subregions. This info is printed into the file balance.out.

Jirka

## ID = 32, Tips for attaining steady state solutions

Hello.
In the hopes of validating our 2D problems with earlier 1D steady-state solutions, I have recently tried using the steady state solution switch. This is activated, I believe, by unchecking the water flow box in the preprocessor's Main submenu. Does anyone have any advice on helping the solver get to a more complete steady state solution?

My problem is a vertical plane of sand cross-cut by a small fault with very different hydraulic properties than the parent sand. My first tests focus on an unfaulted outcrop - sand only. I have tried both top and bottom boundary specified pressures and specified pressure at the top and free drainage at the bottom with uniform initial pressuress slightly wetter than the boundaries, but the solver stops well before the pressure has reached the expected steady state distribution. I even tried setting the pressure and theta tolerances to very low values ($10^{-3}$ and $10^{-6}$, respectively).
John Sigda NM Tech

John,
The option to find steady-state solution to specified boundary conditions (activated by switching off "Water Flow" check box) is not the most robust ones. It succeds mainly for fully saturated conditions, and for unsaturated conditions only when the soil hydraulic parameters are not extremely nonlinear and when the initial condition is reasonable close to the final steady-state solutions.

When HYDRUS-2d fails to find the steady-state solution within specified number of iterations (*5), you will need to run the simulation from time zero until some large time when there are no visible changes in the solution anymore.

Jirka


**ID = 33, Root growth and root distribution parameters**

Dear Hydrus Users,
I have questions regarding to how to account for root growth and root distribution in Hydrus-2D. Does Hydrus allow us to vary the maximum root depth over time (just for 1-dimensional case)since it is appropriate to have increasing root depth over time? Also, what is the parameter "A" in the root distribution parameters block? Referring to the 2 references (Vrugt et.al), do the parameters in that block still reflect the root uptake model by Raats?

Preecha
Preecha,
HYDRUS-2D does not allow time-variable root distribution, i.e., root spatial distribution must be constant with time. The root growth function is available only in HYDRUS-1D.

The command "Parameters for Root Distribution" in the Boundary module has not been part of the original version 2.0 of HYDRUS-2D and therefore this command is described neither in the manual nor in the online help. This command was added when I worked with Jasper Vrugt on his paper for SSSAJ. The parameters for spatial root distribution are fully described in Vrugt et al. (2000).

Vrugt, J. A., J. W. Hopmans, and J. Šimunek, Calibration of a two-dimensional root water uptake model for a sprinkler-irrigated almond tree, Soil Sci. Soc. Am. J., 65(4), 1027-1037, 2001.

Regards, Jirka


**ID = 34, Flowing particles**

Greetings, on the "Graphical Display of Results" page there is a menu choice under "2D Graph" called "Flowing Particles." I do not find any reference to this on the on-line help or in the manual. The choices under "flowing particles" are always grayed out. Is this a functioning option?

Jeff Wright


Jeff,
No this is not a functioning option. It is an option that will be included only into the 3.0 version (and therefore is not described in the current version). Since this version is already under development, some of the new commands are slowly appearing on the menues and in the dialog windows, but are still disabled.
Jirka

Hello, all.
This flowing particles option caught my eye because I am planning a particle tracking analysis of my steady state modeling results. Will this option provide true particle tracking information, such as each particle's path in time, and total time of travel through the domain?
If so, which algorithm do you use to calculate intra-element velocities? Will it accommodate assigning hydraulic properties to elements as well as for the more standard nodal assignment?
If not, have you any recommendations for particle tracking codes which would work with Hydrus-2D output, with or without massaging the data into the right input format?

John Sigda NM Tech

John,
I'm implementing "flowing particles" in the following. At the initial time, user can specify multiple positions as initial position for particles. During the simulation time, these particles are then followed untill final time is reached or they leave the transport domain. Thus, this represent pure convective transport.

I'm using velocities as calculated in HYDRUS-2D, i.e., average nodal gradient (mean of all surrounding elements) multiplied by nodal hydraulic conductivity.
Elemental properties and particle tracking? I have not been thinking about this yet.

Jirka

**ID = 35, Time variable boundary conditions limit?**

Greetings,
Are there any limitations for the number of values that may be introduced for the Time Variable Boundary Conditions. We adopted an hourly time step and simulation is innacurate when the number of values exceed something like 360. Thanks for your reply.

Tamoussi Tamini

No there is no limit on the number of records in the time-variable boundary conditions. The calculation module reads one line at a time and thus the file can be as large as your system, or software allows. The only limitation is within the actual graphical interface. The grid in the dialog windows allows only about 3000 records. However, you can edit directly the atmosph.in file in the folder of particular project, for example using MS Excel.

Jirka

How is it that Hydrus reads the new "in" file? In editing the atmosph.in file, I have created a simulation that exceeds the graphical interface, however, Hydrus still rewrites the input file to emulate what was originally in the graphical interface, and not what I had edited in the text file. Suggestions?

No there is no limit on the number of records in the time-variable boundary conditions. The calculation module reads one line at a time and thus the file can be as large as your system, or software allows. The only limitation is within the actual graphical interface. The grid in the dialog windows allows only about 3000 records. However, you can edit directly the atmosph.in file in the folder of particular project, for example using MS Excel.

Jirka
When you are creating a atmosph.in file outside of the HYDRUS interface, HYDRUS interface must be closed. If you then run HYDRUS interface it will read this new file and when saving data, it will not change the content of the Atmosph.in file.

If, however, you are editing the Atmosph.in file outside of the interface (e.g. Excel)and HYDRUS is running simultaneously, HYDRUS will overwrite your changes when saving data.

Conclussion: When you want to edit the atmosph.in file outside of the interface, do not run it simultaneously.

Jirka

**ID = 36, Floating point error**

Dear Hydrus users,
I have a problem in numerical stability when having large precipitation on the capillary barrier. My barrier consists of large constrast between clay with very low alpha (0.0001 cm-1)and sand with a large value of n (7). I have tried to use different soil hydraulic models and refined my meshes in order of cm. But, the instability still occurs. However, if I reduce the n value from 7 to 5, it could solve the problem. What else could it be done to solve this problem if I want to keep my parameters fixed? I am simulating the field scale (10-m width X 1.5-m deep). I could not reduce my discretization much more smaller due to limitation of cell input.

Preecha

Preecha,
If the program runs for some soil hydraulic properties and not for others, then it can obviously be solved by either spatial or temporal discretization. You need to use finer discretization.
By the way, there is no a big difference between retention curves with n equal either to 5 or 7.

There has been several studies done for capillary bariers with HYDRUS-2D and all of them were rather successful (see some of the references below).

Jirka

Morris, C. E., and J. C. Stormont, Capillary barriers and subtitle D covers: Estimating equivalency, J. Environ. Eng., 123(1), 3-10, 1997.

Kampf, M., and H. Montenegro, On the performance of capillary barriers as landfill cover, Hydrol. And Earth Sci., 4, 925-929, 1997. *

Kampf, M., T. Holfelder, and H. Montenegro, Inspection and numerical simulations of flow processes in capillary barrier cover systems, In:
Holz, K. P., W. Bechteler, S. S. Y. Wang, and M. Kawahara, Advances in Hydro-Science and -Engeneering, Proc. of the 3.rd Int. Conf. On Hydro-Science and -Engineering, Brandenburg University, Cottbus, Germany, pp.13, 1998. *

Mallants, D., G. Volckaert, and J. Marivoet, Sensitivity of protective barrier performance to changes in rainfall rate, Waste Management 19, 467-475, 1999. *

**ID = 37, Too much flow**

Hi All,
I am currently using HYDRUS2D to model water movement in roads. I input rain as an atmospheric boundary condition. I noticed that I could change the maximum allowed pressure at the surface to allow ponding for cases when atmospheric flux into the geometry is more than the hydraulic conductivity. If I don't allow ponding (by letting hcritS=0) or if the head that would potentially build up at the surface is greater than the hcritS I specify, where does the water go? In other words, when I force more water into the medium than its hydraulic properties can handle, what does HYDRUS2D do with this water? Does it just take whatever it can take and then show the discrepancy of the mass balance in balance.out? (I have no flux boundary conditions on the sides)

Defne


Paul,
When you use the Atmopheric boundary condition and the applied flux (precipitation) is larger than the infiltration capacity (Ks for steady-state conditions), HYDRUS switches BC from flux BC to head BC with hCritS as the boundary value. HYDRUS then calculates how much water infiltrates and assumes that all the excess water is immediately removed by the surface runoff.

Jirka


**ID = 38, Solute transport in steady flow**

Hello !

Unchecking water flow (for steady flow study) and checking solute transport, the CPU time becomes extremly long (time step around 0.1) whereas when I only uncheck water flow, the steady state is quickly reached in 4 iterations (initial conditions are close to the final steady state solution).

Does somebody know how to reduce this CPU time, keeping steady flow and solute transport option ?

Julien

Julien,

HYDRUS-2D can directly find steady-state solution only for water flow. It is found for given boundary conditions when "water flow" is unchecked. Solute transport solution is then solved for these water flow steady-state (water content and velocities)conditions. Time step for solute transport is governed by various stability criteria and, thus, depending on given conditions, can be relatively small.

Jirka

**ID = 39, Initial conditions, 2. pressure head**

I am currently using HYDRUS2D to simulate water movement in roads. I have two points in the base layer of the pavement where I have continous water content data from TDR measurements. I select a period of time before which the water content stays almost constant so that I can assume my initial conditions will be at somewhat steady state values. With this assumption, I have tried several different approaches to describe initial conditions and have head varying degrees of success in simulating the temporal water content closely to measured TDR values.

I am wondering if anyone has a suggestion on the best way to describe initial conditions. Using water content values seems to be a good idea (because i know the water content in two locations)however, does not work well because the system becmes far from hydrostatic conditions. In this situation, the water content starts changing as soon as I run the model and I know this is not true because i selected a period of time when the water content was not changing until it rained. On the other hand, if I use the pressure values for initial conditions, I can prescribe a steady state condition but the water content data does not fit what was maesured for the two depths that i have data for. For this option, I keep changing the bottom pressure value and let it equilibriate from the bottom pressure value in the initial conditions meanu. Obviously, all of this becomes more complicated when I don't exactly know what unsaturated parameters I should be using for the model. After all, I am trying to calibrate my model meaning I want to find the best initial conditions and unsat. parameters that will simualte the water content data. I feel confident in the boundary conditions i selected and don't plan to change those.

In short: do you have a suggestion on prescribing initial conditions? I realize that my question may not have a magic simple answer but I just wanted to share my challanges and inquire about other possibilities in case I am missing something about HYDRUS2D.

Finally, I would like to ask another, shorter, question. The pressure head values in the preprocessing and postprocessing menus do not seem to include the elevation pressure. Is that true? In other words, regardless of how I describe my geometry, if i want to have a water table at the bottom of the geometry I should always prescribe a zero pressure head value, is this correct?

Thank you,
Defne Apul

Defne
b) First the easier answer. The displayed pressures, both in pre- and post-processors, are actual pressure heads at particular locations, i.e., without considering z-coordinate. That means that GWL has always a zero pressure, regardless of position.
a) Initial conditions: If your steady-state water contents do not correspond with hydrostatic pressures, that means that most probably your soil hydraulic parameters are not correct and you need to modify them to get the match.

Jirka

Dear Jirka,
Thank you very much for the reply. I have another question related to this. Is it possible to use the inverse option for estimating hydraulic parameters for a steady state condition (i.e. when the water movement box is not checked)? I have tried this for inverse data on water content for two locations but could not get it to work. The message on the DOS window says HYDRUS2D did calculate the steady state conditions but then it crashes because of a division by zero. Does this mean that one can not use inverse parameter estimation for steady state conditions? Is it because the problem too ill posed?
Thank you
defne

Defne,
I would need more information to answer your question. What parameters you try to optimize and how you define the objective function?
Jirka

I am trying to optimize mainly alpha and n but I am also not sure about Ksat, Theta residual, and theta saturated. However, I plan to start by two parameter and once those are optimized, try out the other parameters by inverse optimization. I have three different layers (asphalt concrete, base layer and subgrade). My objective function consists of two water content values for two different locations where I placed observation nodes. Both of these points are in the same layer.

I attempt to find a steady state solution for the system knowing the water content in those two locations. I believe that if I can get the initial conditions to match then I should have an easier time matching time dependent water content data.

In other words, I want to use the steady state inver optimization as one level of calibration and the temporal water content data as a more rigorous way to calibrate.
Defne Apul

Dafne,
From two pointes you certainly can not obtain more than two parameters since there is an infinite number of solutions for such problem if you have more than 2 unknowns. It is like a linera system of equations, i.e., from two equations (two data points) you can obtain only two unknowns {paramaters).

I would not try to use HYDRUS-2D for such problem. If you have two water content data points, and you assume that they are for steady state conditions (thus you should know pressure head), you should use something like RETC code to manipulate tha parameters so that water contents correspond with the pressure heads.

Jirka

dear Jirka and Defne,
I want to add some small comment to this topic (rather late I know).
I also have some problems with the initial conditions. I know my data is quitte good (a sand, which is very offten measured). I know the pF curve of this soil very good but still the adjacent soils and borders have so much effect that the measured watercontent does not match the measured pressure precisely.

Another problem I encountered was when I switch from initial pressure to initial watercontent the original values stay present and are difficult to change. Is this a glitch in teh program?

IJsbrand ,
Now when you mentioned that I noticed that we have some problem when switching from initial condition given in pressure head to water contents. There are several radio buttons, one of which is disabled (hidden) for water contents and it apparently does not work as I would like it to. One can get around it by first changing water content to constant values everywhere, and then editing it futher.

Jirka

Hello everybody,
I also realized that in the Domain.in file the initial condition is allways named "h", no matter whether the initial condition is pressure head or water content. Does this mean anything or is it only a minor nomenclature flaw?

Besides I have another question concerning initial conditions: I'm simulation runoff processes in hillslopes and I'm working with water content as initial condition. Now I tried to simulate a river at the lower end of the domain by defining the boundary in this area as "Constant Pressure" with "Equilibrium from the lowest nodal point". The problem is that I cannot assign values bigger than one to this Constant Pressure BC because if I do it I get the Error message "Initial condition is either smaller than Qr or bigger than QS". Does this mean that, working with water content boundary conditions, I also have to assign water content values to the Constant Pressure BC?

Klaus

Hi Klaus,
Does this mean anything or is it only a minor nomenclature flaw?
This happens with assigning concentrtaions, too.When you click on concentration, the pressure spectrum comes up but when i assign values, i know that they go to concentration not pressure. So, based on my experience, i think it is a nomenclature flaw.

I am not sure if i understand your second question but I know that i used to get the same error anytime my initial conditions had somewhere in the region a water content value higher or lower than the saturated or residual water content for that material. My problem was an artifact of using imported initial conditions that were steady state solutions for a different material.

Defne Apul

hi klaus,
I think you want to simulate a river that has at the borttom a flowcondition of constant pressure (1) =watertable level and the water above has a possitive pressure (also called command) which forces the water into the soil? If not than you should change your boundary conditions. I would try a constant watercontent, you know that if the river is not fluctuating (hight)it is always 100% wetted. Also important that the soil should be in equilibrium with the river at initial conditions. A part of your error message will be propably that the (max)value of the saturated waterconent Qs is exceeded.

Klaus,
The Domain.in file a binary file and thus one cannot read that anything. The ASCII version of this file, i.e., Domain.dat, has some comments lines that are ignored by the program, and one of them indeed always says h for initial condition. It is only a comment line, completely not relevant to the program, and thus I did not see the reason to program several version of it depending on other options.

When you specify initial conditions in terms of water contents, then constant Dirichlet boundary conditions must be also given in terms of water content. This is because constant BC is held in the same vector of values as the initial condition (since it is

never changed during the run), and at the beginning of the run the entire vector is converted from water contents to pressure heads.

Jirka

Hi Jirka,
I gathered that it would be like that but it seemed strange to me to assign a water content to a "pressure"-BC.
Thanks again, your advice is really helpful. I could have saved so much time if I had known of this forum earlier...(of course it would have been your time... ;-)

## ID = 40, Hysteresis

Dear HYDRUS users,
I am trying to include hysteresis in my modeling and have 3 following questions regarding to hysteresis.
1. How does HYDRUS-2D determine the reversal points in order to obtain the scaling factors (alpha) for scanning curves since the reversal points are not required as input parameters in HYDRUS-2D?
2. The Ks(w) parameter is shown in both hysteresis options (retention curve only and retention curve-conductivity). If only hysteresis in retention curve is chosen but Ks(d) is not equal to Ks(w), will I expect to get same results as both hysteresis options are chosen?
3. When lack of data, what is a good approximation for Ks(w)?
Preecha

Preecha,
1. How hysteresis is implemented into HYDRUS-2D is described in detail in the manual on p. 20 through p. 24. There are all the details needed to understant how it is done. The manual is on the HYDRUS CD in the folder "Documents"

2. When only the "hysteresis-in-the-retentin-curve" option is selected, then Ks(w) should be equal to Ks(d). The code will request that from you.

3. When there is a lack of data on hysteresis in the conductivity function, this option should not be used. Actually, people could argue if there is hysteresis in the K(theta) function or not.

Jirka

## ID = 41, A-Level Output File

Dear Hydrus users,
I am simulating free drainage for the bottom boundary conditions for 365 days. The A-level output file only reports the CumQ3, but not CumQ6 for the free drainage, as included in the Cum_Q file. Is there a way to have the A-level file to report the CumQ6 for the free drainage boundary conditions since the P-level time can have

only 100 different time periods? Right now, I just click the t-level information to have Hydrus report every time step, but it does not seem to be practical. The output files can be very large.

Preecha

Dear Hydrus users,
I was mistaking about the p-level information.
Hydrus allows 250, not 100, prescribed time periods, but the time periods are still less than 365 days.
I also would like to check mass balance errors at each day. Is there any other way to do besides rerunning the simulations for different time periods?
Preecha

Preecha,
The CumQ6 information is printed only into the Cum_q.out file. This information is printed either at each time step or at selected print times.
Mass balance information is printed into the balance.out time at selected print times.
The graphical interface will allow you to select maximum 250 print times. However, the latest computational module h2d_calc. exe allows up to 5000 print times. To use that capability, you need to modify the selector.in input file manually using any text editor (you need to keep the formating, i.e. 6 numbers per line).

Jirka

**ID = 43, Q: 2D cross-section tool**

Hello,
I would like to draw a cross-section showing the moisture content profile along the cross-section at several different print times. I am having a couple of problems:
1. Every time I have to graphically indicate the location of the cross-section. Because this must be done graphically, it is difficult to specify the exact same cross-section every time.
2. If the locations I specify for the cross-section end points are outside of the model domain, the distance plotted in the cross-section graph also begins outside the domain, rather than at the domain boundary.
3. After I export cross-section data I am not able to draw another cross-section without exiting Hydrus and restarting the program.

Does anyone have any suggestions for dealing with these issues? Does anyone have a suggestion regarding the best way to plot data from exactly the same cross-section during successive print times?
Eric

Hello Eric:
To your questions:

1. Use grid: menu item "Options->Grid Settings"
2. Unfortunately there is no option to switch off drawing on outer parts of the cross-section. This option might be implemented in next version.
3. This is an error - I'll have to look at it. For now I'd recommend you to export all quantities to a text file (button "export all") and then you can import them to MS Excel (for example) and create your own graphs. I've also noticed that columns in the text file with exported quantities have wrong headings. Columns W1, W2, W3, W4, P, … actually correspond with variables in the combo-box on left dialog bar (Pressure head, etc.).
Best regards Mirek

I tried the grid settings as you suggested but could not see any difference when I tried using the cross-section tool. Am I missing something? I would like to be able to specify the exact same cross-section everytime I use the tool. This is necessary since I can't export data from more than one print time without exiting HYDRUS and restarting the program.
Eric

Hello Eric:
1. "Snap to grid" and "Grid visible" options must be checked and also you should really see the grid. If you don't see the grid then spacing is probably too fine - press button "Default" or enter "reasonable" values for "Width" and "Height". With these options I am able to define a cross-section where I want => I can have same scross-sections for different print-times.

3. My previous suggestion (to export all variables) unfortunately can't be used for different print-times. I'm going to fix this error and will inform you as soon as this problem is solved (my estimation is 1-2 weeks) so that you could download the fixed version.
Regards Mirek

Thanks! Your additional instructions helped me get something I can work with. I look forward to hearing about any modifications you make related to printing the cross-section for several times or the problem I encounter when I export the data and then have to restart hydrus to draw another cross-section.
-Eric

Hello Eric:
I've found the problem and fixed it, however it will take couple of days to prepare a new installation on our WEB site. There is one simple solution how to get around this problem: If you export data to the directory where Hydrus2D is installed then graph component should work correctly (you needn't restart Hydrus2D). After you export all data then you can move created files where you want.

Regards Mirek

The problem has been fixed in version 2.007 that can be downloaded now on our "Hydrus Downloads" page.
Regards Mirek


## ID = 45, Hydraulic properties

Hello Hydrus users,
I'm researcher in subsurface drainage modelling and I regularly use hydrus to simulate the WT fluctuations and drainage discharges when water comes from irrigation.

I wonder if it is possible in Hydrus to dissociate the retention curve parameters from the non saturated hydraulic conductivity parameters. In the VG based on the Mulaem theory, these functions are linked by the same parameters (alpha, n). I would like to use separate parameters (for ex, alpha1, n1 for retention curve and alpha2, n2 for K). Is it possible ?

Thank you in advance,
S Bouarfa


Dear Bouarfa:
This is a question for Jirka Simunek but he is currently in China and I'm afraid he doesn't have access to this forum now. He will be back in 2 weeks.
Best regards Mirek


Dear S. Bouarfa,
HYDRUS does not allow decoupling of the hydraulic conductivity function from the retention curve. To do that would require some modifications in the source code of the computational module. The amount of work (code writing) would depend on how complex the hydraulic conductivity function is. If the new function had only two parameters (e.g. Ks and a shape parameter) this change would be rather straigtforward. If more parameters are needed, then it would be more work.

Jirka


Thank you for your answer.
Yes, I do not need a specific or sophisticate function (Ks and shape function would be perfect). In this case, would it possible for someone from your institute to help us and make this modification in the source?
Thank you in advance,
S Bouarfa


Dear Sami,
Unfortunately I will not have time in the next two month. I have couple of short courses to teach and some other travel. However, you can contact PC-Progress, they

offer similar services on their web page (http://www.pc-progress.cz/Fr_Services_Hydrus.htm), service H2D-B.

Regards,
Jirka


OK,
Thank you for your answer. I will contact them.
Best cordial regards,
S Bouarfa

**ID = 46, How to convert "Domain.in" into ACSII file ??**

Hello Hydrus users,
This is my first time to use Hydrus. I want to use Hydrus to modeling the heterogeneous case. I need to change the material numbers in "Domain.in", but i don't know how to convert this binary file into ACSII file. Anyone can tell me how do that???

Thanks
pringles


Dear Pringles:
in the "Boundary Conditions Editor" go to menu -> "File" -> "Transform to ASCII". This command will create "Domain.dat" - a text file that you can edit. The file Domain.in will be deleted and calculation will use Domain.dat.

Regards Mirek


**ID = 47, Inverse output**

I am trying to use inverse parameter optimization to get a better handle on my unsaturated hydraulic parameters. I have two questions:

1) When using the inverse routine does HYDRUS2D record all the parameters that it runs forward? I know that it records the SSQ, but I can't tell what parameters each SSQ corresponds to. I am interested in this info because I want to better characterize the comparison between the HYDRUS2D output and the measured data and I want to know in greater detail which parameters don't work.

2) Frequently, HYDRUS2D crashes when I use the inverse optimization. However, I can't find where my error is. I try different initial values or reduce the number of parameters being optimized (e.g. from 4 to 3) without really knowing if it will work. Is there a reason inverse optimization would prompt hydrus2d to crash when it would not when it was running forward only once? Has anyone else experience crashing problems using the inverse optimization?
Defne

Ooops sorry. I take back my first question. Parameter values are shown right beside the SSQ coumn. I still have a prb with crashing unless I try to optimize only two parameters at a time. Defne
Defne Apul

Defne,
It is a good idea to always provide some reasonable limits (constraints) on the optimized parameters. The optimization routine (Marquardt-Levenberg) does not check whether new parameters are physically reasonable. If the are not, that could result in crash.

Jirka

Jirka,
Thanks for the answer. I do put boundaries but perhaps my boundaries are not that physically meaningful. I don't have a very good feel for bounding alpha (and somewhat n). It might also just be that I am trying to optimize too many parameters. Thank you,
Defne

**ID = 48, Other inverse questions**

Hi All,

I am using the inverse optimization module of HYDRUS2D to estimate unsat parameters (two at a time) in the pavement.

My inverse runs take only two iterations for estimating two hydraulic parameters. Is this normal: just two iterations and an estimate based on that? I also get fairly small confidence intervals making me think that the parameters were well optimized. However three things are bothering me:

1) my initial guesses come out really close to what HYDRUS2D thinks are the best fit parameters
2) two iterations seem too few to decide that a minimum on the curve is reached (i may be missing the math here). Why does HYDRUS2D stop after two iterations? What is HYDRUS2D criteria for stopping iterations?
2) i try two other values for my initial guesses of the parameters and the inverse optimizations still yield very similar values to my initial guesses. I suspect that there are that many local minimums on the curve. Could there be something else going on? HOw can I have various initial values resulting in the parameter estimates the same as initial values (with very narrow confidence intervals) no matter what the initial value is?

Defne

Defne,
It is indeed uncommon that HYDRUS would converge after only two iterations (see examples, either for HYDRUS-2D or HYDRUS-1D). It seems that your data are such that the objective function is basically flat and does not change much with optimized parameters, i.e., is insensitive to the optimized parameters. Did you review the data and their fit? Is it more or less the same for different values of optimized parameters? HYDRUS stops when it cannot decrease the value of the objective function. Try to optimize only one parameter (i.e., Ks) and see what happens. It is also possible that your initial values are unrealistic and the code cannot find the way towards the minimum, since you are in the flat region of the objective function.

Jirka


I did try it with one parameter and I do get two iterations and then the same input as I put in. I guess, like you said, I do have a flat objective function. I think my input parameters are not on the extreme edges, so that is probably not the reason for too few iterations.
Thank you.
defne


hi dephne
Can you make an initial value that is further away from the first and see what happens then ?
IJsbrand


Hi All,
I am using the inverse optimization module of HYDRUS2D to estimate unsat parameters (two at a time) in the pavement.

My inverse runs take only two iterations for estimating two hydraulic parameters. Is this normal: just two iterations and an estimate based on that? I also get fairly small confidence intervals making me think that the parameters were well optimized. However three things are bothering me:

1) my initial guesses come out really close to what HYDRUS2D thinks are the best fit parameters
2) two iterations seem too few to decide that a minimum on the curve is reached (i may be missing the math here). Why does HYDRUS2D stop after two iterations? What is HYDRUS2D criteria for stopping iterations?
2) i try two other values for my initial guesses of the parameters and the inverse optimizations still yield very similar values to my initial guesses. I suspect that there are that many local minimums on the curve. Could there be something else going on? HOw can I have various initial values resulting in the parameter estimates the same as initial values (with very narrow confidence intervals) no matter what the initial value is?

defne

Hi IJsbrand,
That is exactly what I had tried. I got the same output. As Jirka suggested, I think I have a flat objective function for this problem. I haven't gone back to this problem for a while now. But when I go back to it, will try optimizing one at a time.
Defne

Defne,
Check also the Fit.out output file, that contain the results of the inverse simulation (Displayed in the interface with command "Inverse solution information"). You should look at the end of the file where the information about observed, as well as fitted, values is printed and compared. Make sure that fitted values are not zero. That would mean that HYDRUS did not understand your definition of the objective function (observed data) and could not compare them with simulations. Then, obviously, the same value of objective function would be obtained for any set of parameters.

Jirka

## ID = 49, Solute transport-temperature dependence

Hello everyone,
did anybody already use the temperature dependence for the transport parameters? I tried to use this option for diffusion in the gas phase and the Henry coefficient. The values for the activation energy are supposed to be in ML2T-2M-1 or as stated elsewhere in the online help in J/mol. Do I have to enter the activation energy in ML2T-2M-1, which would mean kg*m2/(d2*mol) since I am using as length unit meters and time unit day or is this independent of the used units and does it have to be in J/mol? If I enter the activation energy in J/mol while doing the calculations in meters and days the simulation crashes with very high Peclet numbers. I used 30000 J/mol for the activation energy for the Henry coefficient and 4500 J/mol for the diffusion coefficient in the gas phase. Are these values reasonable?

Heinke Stöfen

Heinke,
Any particular transport or reaction coefficient is mutliplied by exp[A*(T-T_0)/R/T/T_0,]where R is a universal gas constant (8.314), T is temperature in C, T_0= 293.15K, and A is activation energy in J/mol. I do not know values for Henry's constant for your particular solute, but the order seems right. Try to run the simulation first without temperature effect. Also, you need to have some dispersion or diffusion in the liquid phase, since liquid concentration is the major variable.

Jirka

**ID = 50, Invaild point errror**

Hello everyone,
I've been using Hydrus-2D for 3 weeks now ,things were going great at first ,but now i am having this difficulty:
I am getting an Invalid point Error when i try to run the program. It shows that an error occured when generating the materials..
what is the cause of that :
1- Initial conditions(pressure head or water content)
2- Boundary conditions (constant pressure head or seepage face)
3- When you assign a pressure head initial condition,Can you then assign a constant pressure head within the geometry for some specific nodes?
I would appreciate any information replied on my questions,cause for the past week ,i wasn't able to run even a very simple stimulation.

thanks all.

The initial condition can be given either as the water content or the pressure head. It must, however, be given as pressure head if one uses the constant head BC (since this info goes to the initial pressure head vector).

**ID = 51, Starting HYDRUS2D from visual basic**

Dear All,
I am trying to write visual basic code to automate my hydrus2d. My goal is to change the selector.in automatically and then run hydrus2d. I can do the first task but when i am ready to start hydrus2d I get an error. I know that for batch simulations using run.bat, one executes the h2d_calc. When I execute h2d_calc from visual basic the hydrus2d window says "Open file error in file: Level_01.dir Press enter to continue". If I click on h2d_calc, hydrus2d runs without a problem, so i do know that my level_01 is OK. Can anyone help me figure out what the problem is? Is there more to executing h2d_calc? Also, what exactly is the difference between h2d_calc and h2d_clci? Is h2d_clci executed if an inverse solution is simulated?
I would appreciate any help.
Thank you.
Defne

Defne:
1/ you have to run H2d_calc.exe with a command line argument. The argument is a path to your project, for example "J:\Program Files\USSL\Hydrus2D\Direct\Furrow". If you run Hydrus without the argument then it looks for "Level_01.dir" file that really doesn't exist. But be careful - the path can contain spaces and then it could be considered as several different arguments (=> incorrect path). Look at VB documentation how to run programs with such command line arguments.

2/ h2d_clci.exe is really the inverse solution solver.

Regards Mirek


Thanks for your quick reply Mirek. I will look at VB documentation to figure it out. So, is it true that if I am running the inverse simulation (e.g. in a dos batch file) I should type in h2d_clci <return.txt instead of h2d_calc <return.txt? In inverse simulations does h2d_calc somehow call for h2d_clci or do I need to actually run h2d_clci and not worry about h2d_calc for inverse simulations.
Regards,
Defne
Defne,
In inverse simulations one does execute only h2d_clci.exe. You do not need to worry about h2d_calc.exe, which is used only for direct simulations.

Mirek


hi
I made a batch file in DOs that runned fine.
The major problem is that hydrus needs an enter after the run.
If you include the input file plus an file that contains a entersign. You can work around this problem to. You can also abuse (Make it!)the file it is looking for to make the right simulation. It is justy a text file with the location of the files writen in it.


Dear All,
I am trying to write visual basic code to automate my hydrus2d. My goal is to change the selector.in automatically and then run hydrus2d. I can do the first task but when i am ready to start hydrus2d I get an error. I know that for batch simulations using run.bat, one executes the h2d_calc. When I execute h2d_calc from visual basic the hydrus2d window says "Open file error in file: Level_01.dir Press enter to continue". If I click on h2d_calc, hydrus2d runs without a problem, so i do know that my level_01 is OK. Can anyone help me figure out what the problem is? Is there more to executing h2d_calc? Also, what exactly is the difference between h2d_calc and h2d_clci? Is h2d_clci executed if an inverse solution is simulated?
I would appreciate any help.
Thank you.
Defne


Hi IJsbrand,
I too can run batch files in DOS. Can you be more specific about what you described? I don't think you need to do any tricks like that for running HYDRUS2D in DOS? I just did what is described in FAQ on the PC-Progress website.

Defne

**ID = 52, Error**

Hello All,
Does anyone know what is : Error in IADMAKE
i am not sure why i am getting this error message when i am runing the program..
I hope that any of you have an idea about that ...Please reply asap
thanks.

Hi,
the IADMAKE is a subroutine that generates the adjacency matrix for nodes from the element indidence matrix. Could you specify what problem you want to solve and/or send us your input files?
Regards Mirek

**ID = 53, Flux output**

I have performed a run of HYDRUS2D that simulates one-dimensional infiltration. The 7300-day simulation requires several hundred thousand time steps. I can use the post-processor to generate a graph of "Free or Deep Drainage Boundary Flux" versus time. I would like to generate an ASCI file of time versus the same flux values used to genterate the plot. I have looked at the output files and am not able to locate this information. There is a file of cumulative flow, but the values do not have sufficient precision to back-calculate the fluxes that are apparently used to generate the flux plot.

I would appreciate your quidence on how I can extract this information
from the HYDRUS2D output files.

Hi,
There is one file (Cum_Q.out) that contains cumulative fluxes and one (V_Mean.out) that gives the actual fluxes. You should find your information there. There is an additional file, atmosph.in, that gives selected cumulative fluxes on a daily bases.

Jirka

**ID = 54, Internal flux**

Greetings all --
I need to quantify downward flux through a multilayered variably saturated flow system. The area of interest is at the bottom of a buried trench, and the flux thus needs to be evaluated across the entire base of the trench. This trench ends at the upper boundary (atmospheric BC), and extends to a depth considerably above the base of the modeling domain. As I understand the balance.out file, the InFlow parameter

provides the mass into/out of a region of interest, so in my case InFlow will represent fluxes due to the atmospheric BC, the downward flow out of the region and any lateral flow. This makes my analysis more difficult because of the dynamism in the upper part of the region near the boundaries. I was thinking to establish another region, within the trench itself and far enough away from the top and bottom of the actual trench; however, this might translate to a 0.0 InFlow value (since flow will be near equlibrium through the region), and this won't help either. Has anybody tried to do this using some nice tricks? Also, I am particularly interested in being able to get the flux data at A-Level time periods, not the output times. Jirka, is there a way to do this without recoding anything? Thanks for your feedback.

Regards, Michael

Mike,
There is no simple way. The inflow/outflow is calculated simply by taking mass balance at previous and current time step and dividing by delta t, and thus provide information about the entire subregion due to all fluxes, including BCs, sinks/sources, etc. You are correct that if you look at this value that it includes, in your case, atmospheric BC, lateral fluxes, as well as downward flux. Thus if you create a subregion, that include the trench, you should be able to get, after substracting atmospheric BC flux, the outflow from the trench that includes downward and lateral fluxes. To separate from this downward flux is probably imposible in the currect version.

I have been thinking about calculating flux across internal lines, but have not done that yet. I have hardcoded that option for specific examples, but have not done that for general case yet.

Jirka

**ID = 55, Time step**

Hi everyone,
I noticed that time step has an important effect on runining a program, Is it always better to have a smaller time step to ensure accuracy?
I am dealing with a program that runs at high time step and at low one it gives error, can the results be trusted?
Is time step to be selected according to the accuracy required or it has other relation to the program run.

thanks All.

HYDRUS-2D always automaticaly selects an optimum time step for given problem. So it is good to give HYDRUS a freedom to do that, i.e., by selecting a relatively small initial time step (10s), a small minimum time step (0.1s), and a large maximum time step (depends on application and can be even years). Maximum time step is not important and a very large number can be used. The code will not use that opportunity if the conditions for that are not right anyway.

The precision depends more on iteration criteria than on selected time step. The actual time step plays a role only if the iteration criteria are not selected correctly. The pressure head criterion should be between 1 mm and 1 cm, the water content criterion between 0.0001 and 0.001.

Jirka

## ID = 56, Tortuosity

Dear All,
It seems HYDRUS2D calculates tortuosity based on water content and saturated water content (Millington and Quirk; 1961). Is this the water content of a specific material at a given time? Is it true that the value for tortuosity would temporally change based on how much water there is in a particular node?

I would like to use my own constant value for tortuosity. Is it possible to specify a tortousity value specific to a problem?

Defne

Defne,
The tortuosity is calculated using the Millington and Quirk (1961) formula. i.e., $tau=theta^{(7/3)}/Theta\_s^2$. This means that tortuosity is dynamically changing corresponding with the actual water content (theta) at given time and location.

If you do not want to use this formula and have tortuosity constant, you can simply disable tortuosity factor in the "Solute transport - general information" dialog window and adjust your diffusion coefficient accordingly (since $D=D\_0*tau$).

Jirka

## ID = 57, A bug variable head boundary conditions (kode = 3)

Carrying out some simulations on leaks I head to prescribe variable head boundary conditions (kode = 3) only to one node. Doing so I found something peculiar: The width of the variable head boundary node was set to 0 in the boundary.in file. This lead to system failure as soon as a print out time arrived. Apparently a division with 0 occurs. Interestingly, as soon as 2 variable head boundary condition nodes are assigned, HYDRUS2D (Vers. 2.006) writes out two correct widths for the corresponding nodes.

Sincerily
Hector

Hector,
I have realized this some time ago that the interface can not consider boundary conditions for individual nodes and that there need to be at least two neighbouring nodes on the boundary with the specified BC. This is because the code assigns only the length between nodes with specified BC as a boundary edge (half to each node = "Boundary Width associated with a node"). The reason for that was that we did not want, for example, for the surface corner nodes to include into "Width" also the part of the boundary on vertical sides.

I have modified since then the interface so that it can handle also singular BC nodes, but we have not updated yet the latest download with this. We will let you know once we do that. Since then an easy fix is to rewrite "Zero" in the Boundary.in file with "One".

Jirka


### ID = 58, Dry slope attached to water level - REVISITED

Dear Hector,
have been struggling last days in developing a more realistic boundary conditions for the dynamic interface between the stream (variable head BC) and dry slope attached to water level (seepage face).

I want to assign the nodes below the time varying water level the time variable hydrostatic boundary condition (Kode=3, from the atmos.in file). Above the phreatic level I want to handle the boundary nodes as seepage face. Therefore I first assign
Headnew(n)= 0. and Kode=+13
and let it iterate under the condition:

q(n)=0., Kode=-13 if inflow occurs (q(n)>0).
Headnew(n)= 0. and Kode=+13 if head > 0.

These is exactly the same concept as used in the application of seepage face. However after a while yielding quite consistent results with neat zone of zero pressure above the river water level and switching to no-flux bc above that zone, an obscure floating point exception error occurs.

I suppose, the changing of flux to no-flux in the Q-vector leads to problems, but cannot figure out what really happens. IS the kode=+-13 pointer problematic?

Hector


Dear Hector,

As far as I know the HYDRUS-2D software, there is no Kode=+-13 used. Actually several vector fields in the Fortran calculation module would overflow for such Kode,

since they are dimensioned only up to 6. These must be your modifications of the code and we cannot easily figure out what may go wrong in such case.

I have been actually developing such more dynamic boundary conditions and we are planning to make it available in the version 3.0 of HYDRUS-2D. One of these new BCs was a dynamic change between variable pressure head and seepage face, as can happen at the side of the stream with variable water level. In the test examples I have done I did not encounter any major problems.

Jirka

Dear Jirka,
thank you for your reply. I introduced the Kode(n)=|13| modifications in the code just to point which kind of condition is actually assigned to the node.
Since Kode(n) is an integer vector, I thought any integer number could be appropiate. I will check if there is any command performing an operation depending on the value of icode at a vector which is normally dimensioned up to 6. In fact the exit error encountered is a floating point overflow, however this happens after a while, in which calculations behave physically based.

Hector


Hector,
For example in the TLInf subroutine there is the following cycle:

```
do 12 i=1,NumBP
n=KXB(i)
j=iabs(Kode(n))
if(j.eq.0) goto 12
if(SWidth(j).gt.0) then
hMean(j)=hMean(j)+hNew(n)*Width(i)/SWidth(j)
if(j.eq.4) vMean(j)=vMean(j)-Q(n)/SWidth(j)
end if
12 continue
```

Vectors Swidth, hMean, and vMean have dimensions of NumKD, which is equal to 6. Thus, if you specify Kode=13, there should be overflow.

Jirka


**ID = 59, Is it possible to model ponded conditions?**

Hello all:
I am brand-new to the list. I was wondering if it is possible to model ponded conditions at the top boundary with positive hydraulic head such as 1-2 cm and a zero hydraulic head at the bottom boundary (water table). The initial conditions is dry such as -1000cm throughout the domain except for top and bottom nodes.

This is rather a theoretical question due to the nature of the equation. To what extend and what level, can saturated conditions be modeled with HYDRUS6. At what point, if any, would the model fail? Would the solution diverge if a few nodes near the top boundary become saturated and go to positive hydraulic heads?

I need your expertise on this matter since I am eventually going to link the variably saturated model with an overland flow model.

Thanks.
Orhan

Just to clarify the above situation, here is a simple run I tried to run.

Lenght of domain ---> 62.0cm
Surface BC ---> +5.0cm
Bottom BC ---> 0.0cm
IC for all nodes ---> -1000.0cm
Material ---> Clay soil from Carsel and Parrish (1988)
theta_residual = 0.068
theta_saturated = 0.38
K_s = 5.55E-5 cm/s
Alpha = 0.008 1/cm
n = 1.09

Apparently, after a certain time level, my solution diverged and unreasonable results obtained. Hope this would clarify my problem a bit... Thanks again...

Orhan

Orhan,
I do not see any reason why HYDRUS should be unable to solve this problem. You can rerun Test1 from Workspace Direct with upper BC of +10 cm and see that it works fine, even when couple of nodes at the surface are under positive pressure. Only I do not recommend using a pure van Genuchten model for heavy textured soils, such as clay. This model provides unrealistic prediction of the hydraulic conductivity. Use instead van Genuchten model with the option for a -2 cm air entry value.

Best regards
Jirka

OK!. Modified model runs good with clay soil and ponded top BC. But I have another question. Why is the modified model formulation different in HYDRUS6 documentation than the Vogel et al. (2001) paper?

Vogel, T., vanGenuchten, M. and Cislerova, M., 2001. Effect of the shape of the soil hydraulic functions near saturationon variably-saturated flow predictions", Advances in Water Resources, Vol:24, pp.133-144.

The paper requires only hs=-2cm and a theta_m parameter (2 additional parameters to original model) whereas the form in HYDRUS requires 4 additional parameters. Which one would you recommend? Also, do you have any recommendations for these parameters to be used with Carsel and Parrish (1988) database?

Orhan

Orhan,
In the HYDRUS software there are two implementation of the Vogel et al. (2001) model (Modified VG).

a) First one is the "Van Genuchten -Mualem model" "with air entry value of -2 cm". This model has only one additional parameter to the VG model, i.e., h_a (air entry value). From this value the code calculates theta_m (there is a direct relation between these two parameters).

b) The second implementation is "Modifed van Genuchten". Here there are two new parameters in the retention function, i.e., theta_a (smaller or equal to theta_r) and again theta_m (larger or equal to theta_s). This gives more flexibility at both dry and wet ends of the retention curve. The additional two parameters, i.e., theta_K and K_k relate only to the hydraulic conductivity function. Please see the manual about that.

I would not use these modifications except for heavy textured soils.

Jirka

## ID = 60, Axis-symmetric handling of horizontal wells?

Hello out there,
we would like to analyse pressure data gained in response to the drilling of a release well in a clay soil. However the drillings are not vertical but tilted, some almost horizontal.

Formally a rotational symmetric domain is required to model the pressure propagation to the well boring. Since the symmetry axis does not coincide with the gravity -axis, a tilted system should be set up in which the two components (sinus and cosinus) of the gravitational acceleration have to be considered

I have heard of such FE-systems, especially in the field of hill-slope hydrology, where people build up their (regular) meshes in the x/z coordinate system and then consider the inclination through the components of the gravity.

Would this concept also work for tilted rotational symmetric systems.
Could this be an option for HYDRUS2D?

Hector

**ID = 61, More VB questions**

Mirek,
If you may recall i had asked about starting hydrus2d from visual basic.
You had suggested using a command line which was very helpful. I can get
hydrus2d to start if i type the following in a batch file and then run the
batch file from visual basic:
C:\hydrus2d\h2d_calc.exe C:\hydrus2d\mysimulation

If I do this, I can't get the window to disappear as it does when you run
the batchfile with level_01. I tried adding <return.txt to the end of the
batchfile and that did not help.

1. Do you know how i can get hydrus2d to end without me pressing enter
after the simulation OR do you know what else i should add to the run.bat so
the dos window disappears after the run and i am ready to start another run?

2. I have one other question: I can not get hydrus2d to start from visual
basic (i.e. without using a batch file) even though i can do that with other
programs.

The code i use is:
Sub hadicalis()
Dim RetVal
RetVal = Shell("C:\hydrus2d\H2d_calc.exe c:\Program
Files\hydrus2d\deneme")

End Sub

This code works for wordpad or any other program but not fur hydrus2d. Any
suggestions?

Thank you,
Defne


Defne:

1. You can try to redirect standard input/output. Below is a C++ code that
demonstrates how to do it, VB code would be similar. If you run calculation in this
way it will not wait for "Enter" after finishing and it also will write all "screen
information" to the "A_TaskName_Output.txt" file. This can be very useful especially
if you plan to run several tasks overnight.

... (Function that runs H2D_Calc.exe)

// Prepare the CommandLine. The path must be between quotation marks because of
possible spaces

```
CString CmdLine = cAppFileName + " " + "\"" + cCmdLine + "\"";

// Output redirection
CFile OutputFile;
CString cTaskOutputFileName = "A_TaskName_Output.txt"; // Here you should use
a real task name
if      (!OutputFile.Open(cTaskOutputFileName,         CFile::modeCreate      |
CFile::modeReadWrite | CFile::shareDenyNone))
{
AfxMessageBox("Unable to create the output file");
return FALSE;
}

// Sub-Process information
PROCESS_INFORMATION pInfo;
STARTUPINFO sInfo;

sInfo.cb = sizeof(STARTUPINFO);
sInfo.lpReserved = NULL;
sInfo.lpReserved2 = NULL;
sInfo.cbReserved2 = 0;
sInfo.lpDesktop = NULL;
sInfo.lpTitle = cTitle.GetBuffer(cTitle.GetLength());
sInfo.dwFlags = 0;
sInfo.dwFlags = STARTF_USESTDHANDLES;
sInfo.dwX = 0;
sInfo.dwY = 0;
sInfo.dwFillAttribute = 0;
sInfo.wShowWindow = SW_SHOW;
sInfo.hStdInput = GetStdHandle (STD_INPUT_HANDLE);
sInfo.hStdError = GetStdHandle (STD_ERROR_HANDLE);
sInfo.hStdOutput = (HANDLE)OutputFile.m_hFile;

if (!CreateProcess(NULL,
CmdLine.GetBuffer(CmdLine.GetLength()),
NULL,
NULL,
TRUE,
0,
NULL,
NULL,
&sInfo,
&pInfo))
{
PcxOutError(MGG_IDS_ERR_EXEC, cAppFileName);
return FALSE;
}

OutputFile.Close();
```

// Give the process time to execute and finish
WaitForSingleObject(pInfo.hProcess, INFINITE);
...

2/ I think the problem could be in spaces in the file path. Try to calculate a project with a path without spaces (for example C:\Projects\Test1). If you succeed and if you want to fix your current code try to put the path between quotation marks (see the C++ code above). If this is not the problem then I'd recommend you to run H2DCalc.exe exactly as we do it (see C++ code above). You can use the "CreateProcess" function in VB because it is a general WIN32 API function.

Regards Mirek

Here is an complete example how to spawn a console application and redirect its standard input/output using anonymous pipes.

```
#include <windows.h>
#include <stdio.h>
#include <conio.h>
#include <string.h>
#pragma hdrstop
#include <condefs.h>

#define bzero(a) memset(a,0,sizeof(a)) //easier -- shortcut

bool IsWinNT() //check if we're running NT
{
OSVERSIONINFO osv;
osv.dwOSVersionInfoSize = sizeof(osv);
GetVersionEx(&osv);
return (osv.dwPlatformId == VER_PLATFORM_WIN32_NT);
}

void ErrorMessage(char *str) //display detailed error info
{
LPVOID msg;
FormatMessage(
FORMAT_MESSAGE_ALLOCATE_BUFFER                                    |
FORMAT_MESSAGE_FROM_SYSTEM,
NULL,
GetLastError(),
MAKELANGID(LANG_NEUTRAL, SUBLANG_DEFAULT), // Default language
(LPTSTR) &msg,
0,
NULL
);
printf("%s: %s\n",str,msg);
LocalFree(msg);
}
```

```
//-------------------------------------------------------------------------
void main()
{
char buf[1024]; //i/o buffer

STARTUPINFO si;
SECURITY_ATTRIBUTES sa;
SECURITY_DESCRIPTOR sd; //security information for pipes
PROCESS_INFORMATION pi;
HANDLE newstdin,newstdout,read_stdout,write_stdin; //pipe handles

if (IsWinNT()) //initialize security descriptor (Windows NT)
{
InitializeSecurityDescriptor(&sd,SECURITY_DESCRIPTOR_REVISION);
SetSecurityDescriptorDacl(&sd, true, NULL, false);
sa.lpSecurityDescriptor = &sd;
}
else sa.lpSecurityDescriptor = NULL;
sa.nLength = sizeof(SECURITY_ATTRIBUTES);
sa.bInheritHandle = true; //allow inheritable handles

if (!CreatePipe(&newstdin,&write_stdin,&sa,0)) //create stdin pipe
{
ErrorMessage("CreatePipe");
getch();
return;
}
if (!CreatePipe(&read_stdout,&newstdout,&sa,0)) //create stdout pipe
{
ErrorMessage("CreatePipe");
getch();
CloseHandle(newstdin);
CloseHandle(write_stdin);
return;
}

GetStartupInfo(&si); //set startupinfo for the spawned process
/*
The dwFlags member tells CreateProcess how to make the process.
STARTF_USESTDHANDLES      validates      the      hStd*      members.
STARTF_USESHOWWINDOW
validates the wShowWindow member.
*/
si.dwFlags = STARTF_USESTDHANDLES|STARTF_USESHOWWINDOW;
si.wShowWindow = SW_HIDE;
si.hStdOutput = newstdout;
si.hStdError = newstdout; //set the new handles for the child process
si.hStdInput = newstdin;
char app_spawn[] = "d:\\winnt\\system32\\cmd.exe"; //sample, modify for your
//system
```

```
//spawn the child process
if
(!CreateProcess(app_spawn,NULL,NULL,NULL,TRUE,CREATE_NEW_CONSOL
E,
NULL,NULL,&si,&pi))
{
ErrorMessage("CreateProcess");
getch();
CloseHandle(newstdin);
CloseHandle(newstdout);
CloseHandle(read_stdout);
CloseHandle(write_stdin);
return;
}

unsigned long exit=0; //process exit code
unsigned long bread; //bytes read
unsigned long avail; //bytes available

bzero(buf);
for(;;) //main program loop
{
GetExitCodeProcess(pi.hProcess,&exit); //while the process is running
if (exit != STILL_ACTIVE)
break;
PeekNamedPipe(read_stdout,buf,1023,&bread,&avail,NULL);
//check to see if there is any data to read from stdout
if (bread != 0)
{
bzero(buf);
if (avail > 1023)
{
while (bread >= 1023)
{
ReadFile(read_stdout,buf,1023,&bread,NULL); //read the stdout pipe
printf("%s",buf);
bzero(buf);
}
}
else {
ReadFile(read_stdout,buf,1023,&bread,NULL);
printf("%s",buf);
}
}
if (kbhit()) //check for user input.
{
bzero(buf);
*buf = (char)getche();
//printf("%c",*buf);
```

```
WriteFile(write_stdin,buf,1,&bread,NULL); //send it to stdin
if (*buf == '\r') {
*buf = '\n';
printf("%c",*buf);
WriteFile(write_stdin,buf,1,&bread,NULL); //send an extra newline char,
//if necessary
}
}
}
CloseHandle(pi.hThread);
CloseHandle(pi.hProcess);
CloseHandle(newstdin); //clean stuff up
CloseHandle(newstdout);
CloseHandle(read_stdout);
CloseHandle(write_stdin);
}
```

Dear Mirek and Song,
Thanks very much for your input. I don't know any C++ and currently just a little bit visual basic. However, after your suggestion on redirecting standard input/output, I looked up what this meant, and i was able to put together some visual basic code that does what i needed.

My goal was to get visual basic to (1) read hydraulic parameter data from text file and put it in the appropriate file in hydrus2d, (2)run hydrus2d with these set of parameters, (3)copy the obs.out to a different location and (4)repeat the process. This way i can run many (thousands?) simulations without having to be there. I opted out using a batch file and running DOS because it required pre-preperation of individual .h2d files. With this approach i keep one hydrus2d file and change input parameters, store my output files.

I have spent quite a bit of time on this. For anyone who may need to do something like this or anyone who has suggestions on my code, I pasted below my VB code. Perhaps it may save some people some time:

```
Private Function ExecuteApp(sCmdline As String) As String
Dim proc As PROCESS_INFORMATION, ret As Long
Dim start As STARTUPINFO
Dim sa As SECURITY_ATTRIBUTES
Dim hReadPipe As Long 'The handle used to read from the pipe.
Dim hWritePipe As Long 'The pipe where StdOutput and StdErr will be sent.
Dim sOutput As String
Dim lngBytesRead As Long, sBuffer As String * 256

sa.nLength = Len(sa)
sa.bInheritHandle = True

ret = CreatePipe(hReadPipe, hWritePipe, sa, 0)
```

```
If ret = 0 Then
MsgBox "CreatePipe failed. Error: " & Err.LastDllError
Exit Function
End If
start.cb = Len(start)
start.dwFlags = STARTF_USESTDHANDLES Or STARTF_USESHOWWINDOW
' Redirect the standard output and standard error to the same pipe
start.hStdOutput = hWritePipe
start.hStdError = hWritePipe
start.wShowWindow = SW_HIDE

' Start the shelled application
ret = CreateProcessA(0&, sCmdline, sa, sa, True, NORMAL_PRIORITY_CLASS, _
0&, 0&, start, proc)
If ret = 0 Then
MsgBox "CreateProcess failed. Error: " & Err.LastDllError
Exit Function
End If

' The handle wWritePipe has been inherited by the shelled application
' so we can close it now
CloseHandle hWritePipe

' Read the characters that the shelled application
' has outputted 256 characters at a time
Do
ret = ReadFile(hReadPipe, sBuffer, 256, lngBytesRead, 0&)
sOutput = sOutput & Left$(sBuffer, lngBytesRead)
Loop While ret <> 0 'if ret = 0 then there is no more characters to read

CloseHandle proc.hProcess
CloseHandle proc.hThread
CloseHandle hReadPipe

ExecuteApp = sOutput
End Function
Sub hydrus2dautomationwpipe()
Dim Textline
Dim cAction As String
Dim cfilename As String
Dim retval
sayac = 111

Open "C:\hydrus2d\defne\embankment1D\parametreler.txt" For Input As #2 ' Open
file.
Do While Not EOF(2)
Line Input #2, Textline

Open "c:\hydrus2d\defne\embankment1D\selector.in" For Output As #1
```

Print #1, "Pcp_File_Version=2"
Print #1, "*** BLOCK A: BASIC INFORMATION
****************************************"
Print #1, "heading"
Print #1, "water flow in pavements"
Print #1, "LUnit TUnit MUnit (indicated units are obligatory for all input data)"
Print #1, "m"
Print #1, "days"
Print #1, "mmol"
Print #1, "Kat (0:horizontal plane, 1:axisymmetric vertical flow, 2:vertical plane)"
Print #1, "2"
Print #1, "MaxIt TolTh TolH InitH/W (max. number of iterations and tolerances)"
Print #1, " 20 0.001 0.1 f"
Print #1, "lWat lChem lSink Short Flux lScrn AtmIn lTemp lWTDep lEquil lExtGen lInv"
Print #1, " t f f t f f t f f f t t"
Print #1, "*** BLOCK B: MATERIAL INFORMATION
****************************************"
Print #1, "NMat NLay hTab1 hTabN"
Print #1, " 1 1 1e-008 10000"

Print #1, " Model Hysteresis"
Print #1, " 0 0"
Print #1, " thr ths Alfa n Ks l"
Print #1, Textline
Print #1, "*** BLOCK C: TIME INFORMATION
****************************************"
Print #1, " dt dtMin dtMax DMul DMul2 ItMin ItMax MPL"
Print #1, " 0.001 1e-008 1 1.3 0.7 3 7 100"

Print #1, " tInit tMax"
Print #1, " 0 16"
Print #1, "TPrint(1),TPrint(2),...,TPrint(MPL)"
Print #1, " 0.16 0.32 0.48 0.64 0.8 0.96 "
Print #1, " 1.12 1.28 1.44 1.6 1.76 1.92 "
Print #1, " 2.08 2.24 2.4 2.56 2.72 2.88"
Print #1, " 3.04 3.2 3.36 3.52 3.68 3.84 "
Print #1, " 4 4.16 4.32 4.48 4.64 4.8 "
Print #1, " 4.96 5.12 5.28 5.44 5.6 5.76"
Print #1, " 5.92 6.08 6.24 6.4 6.56 6.72 "
Print #1, " 6.88 7.04 7.2 7.36 7.52 7.68 "
Print #1, " 7.84 8 8.16 8.32 8.48 8.64 "
Print #1, " 8.8 8.96 9.12 9.28 9.44 9.6 "
Print #1, " 9.76 9.92 10.08 10.24 10.4 10.56 "
Print #1, " 10.72 10.88 11.04 11.2 11.36 11.52 "
Print #1, " 11.68 11.84 12 12.16 12.32 12.48 "
Print #1, " 12.64 12.8 12.96 13.12 13.28 13.44 "
Print #1, " 13.6 13.76 13.92 14.08 14.24 14.4"
Print #1, " 14.56 14.72 14.88 15.04 15.2 15.36 "

Print #1, " 15.52 15.68 15.84 16 "
Print #1, "*** END OF INPUT FILE 'SELECTOR.IN' ***********************************"

Close #1
sayac = sayac + 1
txtOutput = ExecuteApp("C:\hydrus2d\h2d_calc.exe" & " C:\hydrus2d\defne\embankment1D")
newname = "C:\hydrus2d\defne\outputs\" & sayac
Name "C:\hydrus2d\defne\embankment1D\obsnod.out" As newname


Loop
Close #2 ' Close

End Sub

Defne Apul


Defne:
Thanks for posting your VB code. I think it can be very helpful to other Hydrus-2D users. Just a little remark: as you can see now there are 2 identical posts from you (the VB code). I don't know why you didn't see your new contributions (may be it was necessary to press the "refresh" button but I'm not sure). I'd like to inform you that you can edit/delete all your posts: login with your name and password and then you can change all your contributions or you can delete them. I could delete your second duplicated post myself (as an administrator) but I'll leave it up to you.
Regards Mirek


Hi,
I just wanted to share with you another way to run sim's one after another. Its very quick and easy (no fancy C/C++/VB programs req'd) - DOS batch files. I have included mine below.

There are two files.
1) do.bat - which is listed below and is the main file doing all the work.

2) enter.dat - this file has just one line w/ enter. To satisfy h2d_calc.exe programs' prompt "Press enter to continue".

Enjoy,
Sevim

--- start: do.bat file ---
@echo off

echo Running simulations. > log.out

```
echo Starting simulation: Test1 >> log.out
h2d_calc Test1 < enter.dat
echo Finished simulation: Test1 >> log.out

echo Starting simulation: Test2 >> log.out
h2d_calc Test2 < enter.dat
echo Finished simulation: Test2 >> log.out

echo Starting simulation: Test3 >> log.out
h2d_calc Test3 < enter.dat
echo Finished simulation: Test3 >> log.out

echo All done. Yippie ! >> log.out
--- end ---
```

Sevim,
This approach is described in our FAQ 27.
Jirka


Hi Jirka,
Thanks for directing me to FAQ site. I have not run into that site until you pointed out. I just joined the forum and am still exploring things about HYDRUS2D. I found this site very helpful.

Thanks again
Sevim


## ID = 62, Time for obs.out

Dear All,
The manual reports that if "short" is set to "true" in Block A (in Selector.in) the output for obs.out will be in preselected print times as opposed to for everytime HYDRUS2D calculated the water flow. However, i could not figure out how and when i select these (P level?) print times. My goal is to get output from obs.out at times i select on the "print information menu". Alternatively, i want to set the times of obs.out to the same level as my precipitation information or other inverse data I have. Am I missing something?

Thank you,
Defne


Dafne,
If you disable in the "Print Information" dialog window the check box "T-Level information", then information is printed into the obsnod.out file (and some others) only at preselected "Print times". These Print times are selected in the same dialog

window. These "Print Times" can obvisouly be set at the same time-level as your precipitation information or other inverse data.

Jirka


Thank you, Jirka. I think it was the "some others" part that confused me. I get it now. This is strecthing it a bit but is there also a way to not have the "some others"? If there is another button or something like that to click, i may find that a big use. I am just asking to see if i can have more uniform output from my simulations but it is also fine with if there is no way: it must be easy to sift through the "some others" using something else.

Defne


Dafne,
By "some others" I meant other output files, namely Cum_Q.out, v_mean.out, h_mean.out, Run_inf.out, and obsnod.out. All these output files are either printed at each time step (when "T_level Information" check box is on, or only at specified print times, whent this check box is off.

Output file a_level.out is written only for time for which time-variable BC is specified.

Other output files (h.out, th.out, temp.out, conc.out, v.out, boundary.out, balance.out) are printed only at selected print times.

Jirka


Dear Jirka,
Despite your online help a while ago I still can't get obsnot.out to print only at selected times. I tried disabling t-level info froom the window menu; i also tried editing selector.in by changing the logical parameter for short to t but neither works. The obsnod.out always prints more time data then what I preselect. Can you think of other things causing this?
I have another relevant question. I want to select more than 100 preselected print times but hydrus2d does not run if it sees a number greater than 100. I tried overcoming this by going into selector.in and specifying MPL as 100 but specifying more preselected times in the rows below. I was doing this for purposes of my uniform obsnod.out file. I haven't yet checked if it works because i can't yet get obsnod.out to print only preselected print times. Are there other options to get around this problem?
Defne Apul


Dafne,
My previous answer was not completely correct. Obsnod.out file is printed (in general release of HYDRUS) at each time level. The other files (Cum_Q.out, v_mean.out,

h_mean.out, and Run_inf.out) are either printed at each time step (when "T_level Information" check box is on, or only at specified print times, whent this check box is off. I have since changed that and now obsnod.out is printed in the similar way. I can send you the executable of that update if you send me an email directly.

More than 100 print times:
The graphical interface will not allow you to enter more than 100 print times. You can nevertheless get around it by going directly into the selector.in file, changing the MPL value, and then entering print time (six values per line). HYDRUS-2D will then display results for all print times. HYDRUS-1D graphics, however is limits how many lines it can show in a single graph.

Jirka

Dear Jirka,
I have a new question related to selected print times now that I can have HYDRUS2D print as many selected times as i wish.

Importing initial concentration and water distribution from a previous simulation is a great feature of HYDRUS2D. Is it also possible to use this feature with times greater than 100? The interface asks for a number less than 100. Is there a way to get around it when the selected print times is for example 350?
Thank you,

Defne

Defne,
The whole program was initially dimensioned to handle 100 print times. When people asked for more print times, we find the way to accomodate that, but did not extend it to the option of importing any of these new print levels. I would need to recompile the entire software and send it to you.

Jirka

## ID = 63, Output files

Hello,
After running my model i've noticed that in the output (cum-Q.out) the cumulative fluxes (cumQ) for two time independent boundaríes (specified head boundary and specified flux boundary)were combined to one flux serie (cumQ1). Is it true that the runoff flux in a very wet period ,when hs is exceeded, is also added to the flux serie cumQ1? Is it possible to split this serie in different series, for each type of boundary one?

regards Gijsbert

Gijsbert,

1) Time variable pressure head and time-variable flux boundary conditions have internally boundary codes +3 and -3, respectively. The program then integrate fluxes over all nodes that have the same (abs(Kode(n)) and prints it into the cum_q.out file. Thus cumulative fluxes for both boundary conditions given above (time variable) are printed under CumQ3. This is the same for constant head (+1) and constant flux (-1) BCs, where it is printed under CumQ1.

2) Runoff is handled only with the atmospheric BC (Kode=+-4). When the infiltration capacity exceeds the potential precipitation, then excess water is removed by runoff. This value (Runoff) is not included into the actual atmospheric flux. That is the reason why potential atmospheric flux can be different from actual atmospheric flux. The other reason is potential and actual evaporation (more common reason).

3) When time variable flux is specified (-3) and the infiltration capacity exceeds this flux, the code tries to enforce the flux, even when it has to impose positive pressures at the boundary. With the atmospheric BC (-4), one can specify the maximum allowed pressure head, from which there is a reduction from the potential to the actual flux.

Jirka


Jirka,

Thank you for your help. I have one other question; is it possible to split the flux through the atmospheric boundary in an evaporation flux, a precipitation and a runoff flux? Instead of only one actual atmosferic flux serie. This because I am interested in the quantity of runoff flux.

regards Gijsbert


Gijsbert,

What code are you using? Is it Hydrus_1D or HYDRUS-2D. I have implemented this flux balances for HYDRUS-1D, but not yet for HYDRUS-2D.

Jirka


Jirka,

I'm using the Hydrus-2D code

gijsbert


**ID = 64, HYDRUS short courses**

This discussion is advertising.


**ID = 65, Comparison w/ HYDRUS 1D**

I am a new user of HYDRUS 2D and had difficulty getting reasonable answers to a simple drainage problem. So I used HYDRUS 1D to get an idea what was going on

with flux leaving the system. The results from HYDFRUS 1D appeared reasonable, so I attempted to used HYDRUS 2D by first constructing a rectangular grid in MESHGEN with the same vertical dimensions and material types as in HYDRUS 1D. I do not get the same results between the two models. Results from HYDRUS 2D are an order of magnitude smaller thasn from HYDRUS 1D. I have reduced the 2D results to a unit width for comparison. Can you help me to understand what might be the cause for the difference??

Smitty:
Jirka Simunek is on the road this week and will be back next Monday. I believe he'll be able to help you...

Regards Mirek
Smitty,
There should be no differences between the two codes (HYDRUS-1D and HDYRUS-2D). Note that when we did verification of HYDRUS-2D that we did multiple comparisons with HYDRUS-1D. For example Test1, Test2, Test4, Test5, and Test6. They all give exactly the same results. Thus, if you get different results from these two codes, there must be some substantial differences in your definition of the problem. Please check that again (boundary conditions, geometry, time and spatial dimensions, etc.

Jirka

## ID = 66, Simple Hydrus-1D obs pt. question

I have a large unsaturated column, through which I am running a variety of pesticide transport simulations. I have inserted 10 observation points using the graphical interface and the model performed with no difficulty. I would like to insert these same exact 10 obs. pts. to all my other model runs (the geometry is the same in each), but is there a quicker way than going into each simulation and inserting these graphically? Could you insert them numerically, similar to how computational nodes can be set?
Best Regards,
Joseph McCarthy

Hello Joseph:
You can edit observation nodes directly in "Profile.dat" text file. Information about observation nodes is at end of this file and consists of two lines: Number of obs. nodes (the 1-st line) and obs. node indexes in "I5" format (the 2-nd line). I assume you have same FEM-mesh in all your simulations - otherwise observation nodes positions could be different...
Regards Mirek

Joseph,
In both HYDRUS-1D and HYDRUS-2D the specification of observation nodes is copied simultaneously with the entire project (i.e., including the finite element mesh).

Thus if you create the geometry once and then just change some parameters, you do not need to reenter observation nodes.

If you, however, want to add observation nodes to the existing project and do not want to do that in the graphical environment then you need to manually modify either the "profile.dat" file for HYDRUS-1D or the "Boundary.in" file for HYDRUS-2D. Information about observation nodes is at the end of Profile.dat and consists of two lines: Number of obs. nodes (the 1-st line) and obs. node indexes in free format (the 2-nd line).

In Boundary.in the info about number of observation nodes is on the forth line and their list is further down in that file. See some of the existing files.

Jirka

**ID = 67, On/Off time**

Hello All,
I would like to know if we have the option to run a simulation for a period of time, followed by a rest time (for ex, allowing drainage to occur), then to continue runing the simulation..

Its like runing a pump at on/off time. Please provide any help on the topic.
Thanks

hallo,
There is certainly an option to vary boundary conditions with time. You can vary pressure heads, as well as fluxes. If, however, you have more complex problem and need to change also the type of BC with time, then you need to run multiple simulations. Run problem with one set of boundary conditions, then prepare a new project with a new set of boundary conditions and import results from the previous simulations, and so on. The good example of that is described in one of our tutorials listed at the ussl web site.

Jirka

**ID = 68, Pressure head**

Hello All,
If I have a saturated column, should the initial conditions be Pressure Head = 0 at all nodes, or should I assign To Pressure head =0 and bottom Pr. Head = depth of the colum (linear distribution with depth)?
The Pressure Head don't take into consideration the elevation head , right?
They gave same result for the column analysis when i tried the mentioned Initial conditions, doesn't seem right to me.

Hi,
This depends on whether you have flux in the column or not. If you want to have flux equal to the saturated hydraulic conductivity, then you should assign the pressure

head everywhere equal to 0. If, however, there is no flux in the column, then you can assign zero pressure head at the soil surface and h="depth of the colum" at the bottom with linear distribution in between. Then your boundary conditions should correspond with this also.

Jirka


## ID = 69, Paddy field

Hello, all
I'm trying to use HYDRUS-2D to estimate transport of water and chemicals paddy field in Japan . But I wonder which boundary condition I should choose.

I'm trying to use Rectangular Geometry (50cm width and 100cm depth)
and ,Water flow BCs are 'Ver. flax' as upper boundary and 'Free drainage' as bottom one.Water flow initial conditions of upper boundary and bottom one are '5cm' and '-100cm'
In 7140 minutes estimaton, heavy rain (over 10mm/hr) is assumed to be fall.
But I cannot find any 'Actual atomospheric Fluxes' in 'water BoundaryFluxes' of 'Post-Processing' collum, even if the water contents of soil did not change.
Where did the water of rain go?


Utchi,
I'm not sure whether HYDRUS-2D will be able to handle properly the boundary conditions you need. HYDRUS-2D does not allow water to accumulate at the soil surface (which I guess is situation common for paddy fields) and it assumes that all water in excess of the infiltration capacity is immediately removed by overland flow.

I would recommend you, as long as your problem is one-dimensional, to use HYDRUS-1D. HYDRUS-1D does have an option that can describe water build up at the soil surface, a situation common for paddy fields. It allows you to start with a water layer at the soil surface, which can slowly infiltrate into the soil depending on the soil hydraulic properties. This water layer can either fully infiltrate with time, or can be "refilled" with a precipitation or irrigation. Thus, HYDRUS-1D may be more appropriate for dynamic boundary conditions needed for paddy fields. This is not possible to do with the current version of HYDRUS-2D. If you know about somebody or some agency that would be willing to fund development of such features also into HYDRUS-2D, let us know.

Jirka


Dear,Jirka
Thank you for your answer.
I'll try also HYDRUS-1D.
But another question arose.
You said that HYDRUS-2D did not allow water to accumulate at the soil surface. But then what does the 'Constant Pressure' mode means ?

I thought that the mode intended accumulation of water.

There are many 'enigma's on options of HYDRUS ,many variaton of boundary conditions is only one example. If you have the list of meaning of boundary conditons, would you introduce us ?

Utchi,
Constant head boundary condition will impose constant head at the boundary. This value can be both positive (water layer) or negative. The code will calculate how much infiltrates with time. The code will keep the boundary head constant, i.e., water layer is independent of amound infiltrated or any precipitation or irrigation events.

Jirka
Jirka,
Thank you for your answer.
'Constant Pressure' means constantly controled water surface level, doesn't it? All right.

Then, I want to back to my first questions ---about that I cannot find any 'Actual atomspheric Fluxes' in 'water Boundary Fluxes' even if it rains.
What do you think of it?

Utchi,
are you sure that you specified not only precipitation flux in the "Time variable boundary condition" window, but also designated part ofthe boundary as atmospheric boundary in the "Boundary" module (Boundary Conditions Editor)?

Jirka

## ID = 70, Is it faster?

Greetings to all,
I am curious if it might take my computer less time to process if i executed hydrus2d as a fortran code as opposed to executing h2d_calc.exe. I am mostly interested in this because I want to run multiple runs, one after the other. I thought keeping the interface out of the execution may speed up my simulations. Is this possible? I would be willing to buy a fortran compiler.
Thank you,
Defne

Hello Defne:
h2d_calc.exe is a Fortan code (with maximum speed optimization) and it will not run faster if you run it without H2D interface. I think there are only two ways how you can speed up your calculations:
1/ Buy a faster processor

2/ If you have a local PC-network you can run your simulations on several PCs simultaneously. All PCs should be able to access a common Hydrus-2D project directory on a shared disk. Of course it is not a real parallel processing (every single PC calculates its own tasks) but it can be very efficient especially if you have 3-10 PCs and it doesn't require any other investments.

Regards Mirek


## ID = 71, Solute transport parameters

Hi all:
Now I am using Hydrus-2d to fit the model solution to miscible-displacement data in order to estimate fate and transport parameters.
I have two questions:

1. Are there some tips to set the Init. Min. and Max values of the solute transport parameters? Because different setup will get different result, my sencond question is:
2. How to know the result is uniqueness or non-uniqueness, in other word, how to get the unique result for the data?


Song,
Questions of uniqueness or non-uniqueness (and/or solution stability, ill-posedness, parameter identifiability, parameter correlations, and goodness-of-fit) are one of the more complex questions concerning parameter estimation technique. This discussion is beyond the space we have in this discussion forum. I suggest that you carefully study the papers I list below that cover this subject. Especially chapters from the Methods of Soil Analyses, recently published book by SSSA may of interest.

Jirka

Hopmans, J. W., and J. Šimùnek, Review of inverse estimation of soil hydraulic properties, in van Genuchten, M. Th., F. J. Leij, and L. Wu (eds.) Characterization and Measurement of the Hydraulic Properties of Unsaturated Porous Media, University of California, Riverside, CA, 643-659, 1999.

Šimùnek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

Hopmans, J. W., J. Šimùnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Šimùnek, J., D. Jacques, J. W. Hopmans, M. Inoue, M. Flury, and M. Th. van Genuchten, Solute Transport During Variably-Saturated Flow - Inverse Methods, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 6.6, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 1435-1449, 2002.

**ID = 72, Floating point error**

Within the framework of my Phd i am doing inverse optimization of tension infiltrometerexperiments. I am using the Hydrus2d model for this. In the paper of Water Resources Research of october 1999 concerning 'estimating unsaturated soil hydraulic properties from laboratory tension disc infiltrometer experiments' it is mentioned it is necessary to include the final water content in the objective function in order to get an unique solution. However doing so in Hydrus2d (including the final water content as a h(theta) measurement in the objective function) always leads to a floating point error, not only with my data but also when i tried to run the examples that come together with the Hydrus2d model (Crust, Disc). Running the model without the h(theta) measurement doesn't give any problem and does fine.

Do you know what is wrong and how i can overcome this problem? I am including the h(theta) measurement in the sheet for 'data for inverse solution' as follows:
X Y Type Pos Weight
-3 0.27 5 1 1


Sophie,
Well, I have found out that since I did these examples, I have modified the code so that these examples could not be run anymore. This was my mistake and I do apologize for that. I'm emailing you a fixed code that should be able to run both examples without any problems (replace it in the HYDRUS2D folder). Thanks for discovering this bug in the code.

Jirka


**ID = 73, Simulating a point source**

Hi All:
I am trying to simulate a emitter with a constant flow rate at the soil surface. So far I have tried by using a nodal recharge/discharge and it seems to work OK but when I checked the plots I realized that I have a considerable pressure (about 5 cm) at the nodal recharge/discharge. I really wasn't expecting this result since the ponded area shouldn't be producing this pressure (The ponded area is a circle with radius between 6 and 7 cm). My question is... Is there any other way to this and getting the right result? meaning no presure at the nodal recharge/discharge?. I would appreciate any help.
Thanks!
Mat314


Mat314,
When you specify nodal recharge as boundary condition, the code will try to get the flux in the profile even if it needs to apply positive pressure. Your nodal recharge was probably larger than the saturated hydraulic coductivity and then you need a positive

pressure to get water in. You can divide this flux into two or more nodes to avoid positive pressures. I'm working at this moment with Jan Hopmans on simulating (increasing) ponding from drippers but that option is not yet available.

Jirka


Hi Jirka:
I will be waiting for the option to simulate an increasing ponding area from an emitter since that is exactly what I am working on. In the meantime, since that option is not available I will take your advise and try to work using nodal recharge/discharge in more than one node hoping to get better results. Thanks again!

Mat314


## ID = 74, Vertical TDR

Dear all,
I am interested in doing inverse optimization on moisture content readings from vertically installed TDR probes. However, using hydrus-2d, you can only enter moisture content as a function of time assigned to a certain observation node and not as the average over several observation nodes. I was wondering if the program could be adapted to do so, or can it be coupled with another routine or so?
Can anybody help me with this problem?

sofie herman
lab for soil and water
Catholic univeristy Leuven


Sophie,
I have encountered similar problem in the past and implemented a solution into HYDRUS-2D. Since it was not in the original release this option is not documented. If you have one of the later version of HYDRUS-2D (such as the one which you can download from this site) you can enter into the objective function the average water content over selected subregion. To do that you need to enter Time (X variable), Average water content (Y variable), type (2), Position (-subregion number), weight. Thus you can see that the only difference between specifying water content in the observation node or average water content in the given volume is the sign of the position variable.

Jirka


Dear Jirka,
I tried to run Hydrus-2d including the vertical TDR probe readings into the objective function. Hydrus-2d ran very well. I got an SSQ for measurement type 2, so i think it worked. Now i would like to know if i can compare the measurements with the fits?

In fit.out, i can't find the data from measurement set 2 together with the predicted values. Is that normal?

sofie herman


Sofie,
There was a bug in the program so that everything was working fine, except that the average water contents were not printed into the fit.out file summarizing the fit to the objective function. You could still find this information in the balance.out file though. I'll email you an update.

Jirka
## ID = 75, CXTFIT/STANMOD

My name is Jean-Baptiste Dessogne and I am student in post-graduate diploma (equivalent for "DEA") at the University of Burgundy (France).
My work consists in modeling downward transfer of pesticides in wine-growing soils and is conducted by the assistant professor Sylvie Dousset (Sylvie.Dousset@u-bourgogne.fr). The model we are going to use is CXTFIT with STANMOD graphic interface (version 2.1 from Toride et al., 1999). Although I understood the global working of the model, I still have some questions about the equations. Would you mind if you could give me some answers?

In the case of deterministic physical non-equilibrium model (without taking into account degradation for now), Stanmod interface requires, as input parameters, average pore water velocity v, dispersion coefficient D, beta and omega (dimensionless mass transfer coefficient). And the same parameters are given after the run in the output text file (all the notations are taken from vanGenuchten and Wierenga, 1976). But it does not give directly other parameters as fraction of mobile water (phi) and fraction of adsorption sites in dynamic region (f). I did not understand very well how I could infer those parameters from those given by Stanmod (i.e. eq3-6 from vanGenuchten and Wierenga, 1977 part III). Besides, eq6 from this publication suggests a non-linear equilibrium adsorption in the mobile domain. But I believed that the model only took into account linear adsorption? Stanmod interface does not require giving a value of theta (water content) in the input parameters. Does the model recalculate it from the value of V or does it calculate a value of theta by another way? Moreover, does the value of theta used by the model refer to the "effective" water content, that is to say the mobile part, or the total water content?

I noticed in vanGenuchten and Wierenga (1977 part III) that you were able to use two different values of K for adsorption and desorption. Is it possible to do the same with the Stanmod interface?

At last, to the best of your knowledge, is there research teams or
laboratories, in the US, with the same research thematics and which
would be interested by postgraduate students? Indeed, as a postgraduate student, I would like to continue my studies with a thesis.

Jean-Baptiste Dessogne

Dear Jean Baptiste:

I got your inquiries (below).

1. The parameters beta and omega are dimensionless parameters. Specifically, beta includes the effects of immobile water (phi) and fraction of equilibrium sorption sites (f), which have the same effects on the transport calculations (beta). You should be able to input them separately. However, you can not fit those two parameters (phi and f) simultaneously since they are correlated (the model cannot tell the difference). You can infer phi and f from beta in two ways:

a. by using two tracers (say chloride, or better tritiated water) and your sorbing chemical, and assuming phi to be the same (two equations for beta with two unknowns).

b. Assume that f=phi, which seems like a reasonable assumption, and has been done in several manuscripts (e.g., Nkedi-Kizza et al.).

2. STANMOD and hence CXTFIT2 are based on analytical solutions, which require equations that are linear in c. They cannot handle nonlinear sorption. In that case you need to go to a numerical solution. I would recommend HYDRUS-1D for that purpose.

3. Theta indirectly is used as an input parameter to calculate v from the Darcian flux, q (v=q/theta), as well as pore volume if dimensionless time is used. Theta refers to the total water content. Good codes and manuals within STANMOD to study first are CFITM and CFITIM. They are easier to understand and the codes are easier to use for break through curves.

4. STANMOD cannot use two different values for K (adsorption and desorption). That would again imply nonlinear sorption (hysteresis).

Given the above questions, I would very strongly urge you to consider using HYDRUS-1D (http://www.ussl.ars.usda.gov/models/hydr1d1.HTM) for your data analysis. HYDRUS-1D is much more flexible by being able to simulate nonlinear nonequilibrium sorption, as well as transient water flow. In the end it may be more advantageous also when you are looking at (or extrapolate to) field-scale processes. This unless you perhaps only study transport in homogeneous soil columns. But even then you could use (and need) HYDRUS-1D for nonlinear sorption.

4. There are many universities and research groups that work on the same problems. Look at the internet. Several, I am sure would be able to use postdocs. I suggest that you contact people directly (e.g., those you recognize from the literature). I could suggest a few locations/people if you want. Unfortunately, our situation here is less then ideal in terms of funding. We are unable to provide any financial help.

Rien van Genuchten
George E. Brown, Jr. Salinity Laboratory, USDA-ARS

**ID = 76, Boundary fluxes**

I am modeling the "draindown" of a tailings impoundment before and after cover placement and have the following questons on the HYDRUS capabilities (all related to output capabilities) :

1) How can I extract instantaneous and cumulative outflow for a user-defined section of the model boundary? The default output options in HYDRUS (Q vs time) only seem to provide the total (net) flow for a given type of boundary condition (say constant pressure or seepage face). This causes problems if I have the same model boundary type at various model boundaries. For example, in the case of a simple column model where a constant pressure is specified for the top and bottom boundaries there seems to be no way to get the inflow into the top and outflow at the base as separate outputs. Do you know of a way to extract such information quickly using HYDRUS or the WMS output files?

2) A similar problem is encountered when I want to know the water (or solute) flux across a user-specified section (line segment) in the interior of the model. Is there a simple way to extract such an nformation from the output files provided by the model?

3) I have also tried to extract information about simulated water flux
across a section using subregions (perpendicular to the direction of low) and looking at the water balance. Unfortunately the water balance only seems to give me only the net flow (=outflow-inflow) across the entire subegion. Is there a way to extract a complete flow budget for a selected region (e.g. all inflows and all outflows as provided by e.g. MODFLOW in its FLOWBUDGET module)?

Christoph Wels Ph.D., M.Sc.
Principal Hydrogeologist
Robertson GeoConsultants Inc.


Christoph,
1) You are right about the reported fluxes. They are only for the entire boundary of a particular type. There is, however, possible to get actual fluxes across any boundary nodes from the output file boundary.out. There you can just select nodes in which you are interested and add Q values (e.g., in Excel). If you need I can add to this file also solute fluxes (Qc).

2) I have developed a special version that can calculate fluxes across specified cross section. This part is however not supported by the interface and user has to manually describe the cross section (nodes and elements at the cross section). In principal this should be applicable also to boundaries, but I have not tested it for that. Let me know if you are interested in this option and I will send you instructions on how to do that.

3) Unfortunately there is not such information and I do not know about simple way of getting it.

Jirka

Dear Christoph and Jirka,
I have been using v.out and conc1.out files to extract flux information across a boundary. I import these files into excell and use macro-visual basic to extract the velocity and concentration at the nodes that I specify. It does this for each time. Then i multiply the concentration with velocity to get solute flux. Christoph if you are interested i would be happy to send you the code, it is pretty basic. Jirka, if you see any problems with this approach that i may not be aware of, would you let me know?

Defne Apul

Defne,
Your approach captures only part of the solute flux. Solute flux is composed of two parts - convective and dispersive flux. Your approach handles only convective flux and ignores dispersive flux. Your results should be more or less the same as reported fluxes on the outflow boundaries, where only convective flux is allowed. However, on inflow boudnaries, or inside of the domain, dispersive flux can be important.

Jirka

## ID = 77, Preferential flow

Hello,
I am interested in trying to assess the impact of preferential flow on solute transport under instationary boundary conditions (modeled infiltration with the atmospheric boundary condition). I want to perform "numerical experiments". Looking at it in 1d will be suffucient.

I have the version 2.01 of Hydrus 1d which obviously supports the use of a dual-porosity medium (two modal retention function), although there is no Help available on this topic yet. I found a paper by Zurmühl and Durner (1996, Modeling transient water and solute transport in a biporous soil, Water Resources Research, Vol. 32, No. 4, 819-829) where they suggested three different methods to treat the immobile water. As far as I can see only one of these three methods is possible in HYDRUS 1d, that is to treat the immobile water content as constant.

Is there a way that I can implement the other two options: to use a constant portion of the water-filled pore space as immobile water content or to define the immobile water content as a dynamically varying property which is defined by a constant ratio of conductivities. If these options are not possible yet, do you plan to implement something like this in the future?

For the assessment of the impact of preferential flow on transport of sorbing contaminants a more mechanistic model of the kinects of sorption would be needed. Is the implementation of an intraparticle diffusion code something you have been thinking of?

Heinke

Hello Heinke:
This is a question for Jirka Simunek. Unfortunately he is not available this week and should be back on Tuesday, February 4-th.
Regards Mirek


Heinke,
We are implementing a multiple approaches that could be used to describe the preferential flow and transport. These approaches include dual-porosity model with either constant or variable immobile water contents, and dual-permeability models either with kinematic wave equation for macropores or with the Gerke-van Genuchten approach with two overlapping media with its own characteristics. Most of this is described in our upcoming paper which you can get from the web site of the Journal of Hydrology.

Šimùnek, J., N. J. Jarvis, M. Th. van Genuchten, and A. Gärdenäs, Nonequilibrium and preferential flow and transport in the vadose zone: review and case study, Journal of Hydrology, in press.

I have not thought about implementing those two additional options as given in Zurmühl and Durner (1996), but that does not seem to be difficult to implement. We can send you the source code of the program in the HYDRUS-1D currently being distributed by IGWMC if you want to do that.

Jirka


## ID = 78, Solute mass increases for no reason

Dear All,
Lately, I have been using HYDRUS2D to simulate leaching of contaminants from coal fly ash when it is used in road construction. I have a 1D design where I have the ash 0.2-0.5 meters deep in my 1m mesh. The Kd values for my equilibrium model is different for the ash and the soil. I specify an initial concentration and don't let any solute come into the region from the atmospheric boundary condition.

I checked my balance.out and noticed that my concentration keeps increasing although there are no sources for new solute once after the initial concentration has been specified. Any ideas on where i might have gone wrong?

I have checked all that i could think of and can't figure why my mass balance for solute does not work.

Defne


Actually, i found out what was wrong. Two engineered materials with very different Kd values needed a much much finer mesh than what I had. The problems was fixed

when i changed my mesh. For anyone interested,this was also causing negative concentrations.
Defne Apul


Dear All,
I take my word back, i do notice negative concentration even when balance.out shows good mass balance on solute. This happens when I run the model for a very long time. (I am running the model for 10 years with quarter hourly precipitation rate and equilibrium sorption.) What does negative concentrations in HYDRUS2D mean?

Defne Apul


**ID = 79, Variable pressure boundary condition**

I'm attempting to simulate a flux of liquid waste from a surface trench into the vadose zone. Liquid infiltration from the trench will occur at different nodes along the trench, and at differing times. I think I want to simulate this by assigning the trench nodes as a variable pressure boundary condition. I'm looking for help on how to assign selected nodes as having a pressure of zero at the various times the selected portions of the trench are filled with liquid.

Currently, the surface nodes are assigned as an atmospheric boundary condition to accommodate time variable rain events.


Hi,
The current version of HYDRUS-2D can handle only one type of boundary condition at any particular boundary node during one simulation. The code does not allow the dynamic development of boundary type, such as from atmospheric BC to variable head and back, as is probably needed for your application (with water level going up and down). You need to do a sequence of runs, when you import final results from one simulations run to the other, and change boundary types between runs.

Jirka


**ID = 80, Graphical display of results**

Hello,
To display my results properly, I have set the increment of the isolines on 0.1. I also want to give the areas between the isolines different colors. However, the color spectrum is fixed on an increment of 0.5. My question is how can I change the color spectrum in such way that it will correspond with the increment of the isolines.

Gijsbert,


Hello Gijsbert:

A/ To set isolines increment go to "Options" -> "Isoline parameters" and input your increment (0.1)

B/ Then go to "Options" -> "Color Palette" and check "Adjust using colors to the spectral scale". May be you will need to use a different palette or to change palette colors. You can also create your own color palettes. However you can't define values of the scale. They are calculated automatically and user-defined scales will be supported in next version.

Regards Mirek


## ID = 81, Cbound1, Cbound2, etc.

Some of you will find this question frustratingly simple. My apologies in advance, I'm a Hydrus2D beginner. How does one define which time independent BC goes with which Cboundx? I know where to enter the value for Cboundx, but how do I define which time independent BC it is being linked to?


cBoundx values are used only with time intependent boundary conditions, ie., constant head or flux BC. The concentration values for time-variable BCs are specified in columns cValue1 through cValue3 in the "Variable Boundary Conditions" dialog window. Which value is used is specified in the "Boudnary Condition Editor" when specifying solute transport BCs.

Jirka


## ID = 82, Parameter estimation, cxtfit

Dear Prof. van Genuchten,
I am a PhD student in the University of Erlangen-Nuernberg, in Germany. Right now I am doing my research work in the sorption of VOCÆs (TCE and 1,2-cis-DCE) on soils with low organic carbon content
(< 0.1%). I am doing my experiments with a soil column equipment, and I started using the software CXTFIT to estimate the parameters like pore water velocity, dispersion and retardation.

I am using KCl as a tracer together with TCE and 1,2-cis-DCE.

>From the breakthrough curve of the KCl, I estimate the pore water velocity and the dispersion coefficient, fixing the retardation to unity. Then I use the breakthrough curve of TCE (or DCE) with the results from the tracer in order to estimate the retardation
factor.

Since I do not know the porosity of the sample, I would like to ask you if it is correct to estimate both parameters at the same time, pore water velocity and dispersion, directly from the tracer experiment.

I would appreciate a lot your help on this matter.
Manuel Alejandro Salaices Avila


Dear Manuel:
Yes, you can estimate D and V from a BTC, i.e., any two of the three CDE parameters (v, D, R), but not all three simultaneously since they are linearly dependent. If you know the applied flux, q, then you can estimate the effective water content (=q/v) (~ porosity if fully saturated). Of course, this assumes that R=1 for your tracer, KCL, which may not be the case if you have anion exclusion (R can easily be 10-20% less than 1.0 if have a fine-textured soil in your column). So why not measure the water content after your experiments? Than you can confirm the R=1 assumption, or change your R to whatever is appropriate.

Once you have the above values (with or without the R=1 assumption), then the TCE transport analysis should be straightforward. Using CXTFIT assumes that you have linear sorption. Have you used the windows-based STANMOD version? CFITM within may be a bot easier to use for breakthrough curves only, with and witout the nonequilibrium assumption. For nonlinear sorption you may need to use HYDRUS-1D.

Rien van Genuchten


Dear Manual,
If you have information on multiple degradation species (e.g., TCE, DCE, ..., ethylene) you should look at the two papers given below. In those we used HYDRUS-1D and HYDRUS-2D to simultaneously analyze multiple species and their transport and degradation properties.

Schaerlaekens, J., D. Mallants, J. Šimùnek, M. Th. van Genuchten, and J. Feyen, Numerical simulation of transport and sequential biodegradation of chlorinated aliphatic hydrocarbons using CHAIN_2D, J. of Hydrological Processes, 13(17), 2847-2859, 1999.

Casey, F. X. M., and J. Šimùnek, Inverse analyses of the transport of chlorinated hydrocarbons subject to sequential transformation reactions, J. of Environ. Quality, 30(4), 1354-1360, 2001.

Jirka


## ID = 83, Observation node

Hi all,
The numbering of the observation points change with zooming in/out, how can I get the right observation node number to check my results.

Hello,
this is a bug. Correct numbers are those that you see in "View All" mode (when all observation nodes are visible). This error will be fixed as soon as possible. I think that a new installation could be ready for downloading on Monday, February 10-th, 2003.
Regards Mirek


## ID = 85, Regarding calculation of deep percolation

I am using HYDRUS-2D for simulating water movement in rice field (On-farm reservoir concept). I need to have the information on how to calculate deep percolation at a particular point below the root zone. I will be happy to know any related reference/formula/procedure using hydrus-2d to calculate this.
expecting co-operation from the forum.

Laxmi Narayan Sethi
Research Scholar
I.I.T., Kharagpur
West Bengal - 721 302
lnstb@rediffmail.com


Laxmi,
HYDRUS reports information about actual and cumulative fluxes for any type of boundary conditions you specify. Thus if you specify, for example, free drainage BC at the bottom of the transport domain, you should get an output about actual and cumulative flux accross this BC in v_mean.out and CumQ.out, respectively. You can display both graphically as well. The values for any particular boundary nodes are reported in the boundary.out file.

If you want information about fluxes in nodes in the domain, then you need to find this node and either:
a) in the "graphical output of results" view velocity vector for this node and using cross-section option get its value, or
b) Convert the output file into an ascii file and find vx and vz components of velocity for that particular node.

Jirka


Dear Jirka,
Thanks alot for your reply to my problem for calculation of deep percolation, but I couldn't solve the problem I need more enquiries. The problems are:
1) in input paramters of assigning deep drainage BC {The reference groundwater level position, Aqh parameter in q(GWL)-relationship and Bqh parameter in q(GWL)-relationship} where I am in doubt what relation and value should be put for simulation.
2) Iteration time ( Maximum no of iteration, Time step controls)
The main aim of mine is to quantify the lateral and vertical deep percolation in rice

fields and the effects on groundwater levels. so, I am requesting you all kindly help me to solve my difficulties in using Hydrus 2D.

Thanking you.
Laxmi Narayan Sethi

Kharagpur,
1) Deep percolation is implemented as functional relation that relates bottom recharge to the position of the groundwater in the soil profile. The discharge rate q(n) assigned to bottom node n is determined by the program as q(n)=q(h), where h is the local value of the pressure head, and q(h) is given by

q(n)=-Aqh*exp(Bgh*abs(h))

where Aqh and Bqh are empirical parameters which must be specified. This function (similarly as "free drainage") should be used only at the bottom of the profile, not sides. See the manual or the reference below.

Hopmans, J. W., and J. N. M. Stricker. 1989. Stochastic analysis of soil water regime in a watershed, J. Hydrol., 105, 57-84.

2) Iteration time ( Maximum no of iteration, Time step controls) - I would recommend to keep the default values as given in HYDRUS. Except perhaps for the minimum time step, that could be about 1s.

Jirka

Dear Jirka,
Thanks for your reply. I am agreed with your reply but the problem is that how to get the values (local pressure, empirical parameters Aqh and Bqh). Those relations I have already found from mannual. I have to simulate the VF and HF for 20 years to determine the optimum size of on-farm reservoir for rice and mustard cropping system. I have only three years of groundwater level data for shallow ( varies from 1 m to 6m) and deep observation wells/ piezometers 9 varies from 16m to 19m from the ground surface). So, I need of your help to get those value. I have collected the references you mentioned in the last. I am still in middle to get that. Ok, lets hope for your reply.

yours sincerely,
Laxmi Narayan Sethi

Dear Laxmi,
Let me refer to the following reference in Agricultural Water Management:

13. Hopmans, J. W. 1988. Treatment of spatially variable groundwater levels in one-dimensional stochastic unsaturated water-flow modeling. Agricultural Water Management 15:19-36.

It is shown in this paper how the coefficients are determined.
In concept, they can only be determined from measurements of groundwater table and drainage discharge of the field. If you have these data, you have a very nicely defined lower boundary condition that provides fluxes that are calibrated from measurements. Likely, the magntitudes of these parameters characterize the drainability (magnitude as a function of time) of the field, and is likely field-specific, depending on my parameters (e.g. drain spacing, soil type, macropores presence, drain material, lateral flow of groundwater, leakage to deeper groundwater, etc.). So, I would be careful in using values that were used by others, unless your data indicate that they might be valid.
Cheers, Jan Hopmans.


Dear Jan Hopmans,
Thanks a lot for your reply. I have collected your mentioned reference. Let me go through that. That paper describes the relation between watershed discharge and groundwater level. The relation subjected to regression analysis to get the parameters. In my study I have 18 plots of rectangular shape (40m ´ 20m) and an on-farm reservoir of size about 10% of the plot size. Shall I consider the runoff generated from the individual plots or all plots with the groundwater level of observation wells (7 shallow and 13 deep) in the study site to get those parameters? I have also the well log of the observation wells. Is it right to consider the reference of clay layer position with the groundwater level as the reference groundwater level position and groundwater level position with mean sea level as absolute groundwater level position? I hope you will make some clarification and needful suggestion to go forward. Let me wait for your reply.
Thanking you.
Laxmi Narayan Sethi


Laxmi,
I must partly guess of your modeling intentions. I assume that you have plots with different treatments, and that the ultimate goal is to predict sil moisture, etc, for each plot.

Preferably, you would like to find a single q-h relationship that will apply to all plots. The watertables (h) might vary among plots because of their different treatments, resulting in variations in discharge among the plots. Thus, I would plot all h-data (but use different symbols for each well) and plot against the overall discharge (that presumably is measured to the storage reservoir). The disharge is relative to all plots combined (large area).

As reference I would use relative to the soil surface if the plot area is generally flat. If you have alot of topography, it may complicate matters alot. Hope this will help. Jan
Dear Jan Hopmans,

Thanks a lot for your reply. Sorry, I couldn't get your answer. My aim is how to get the input paramters of assigning deep drainage BC {The reference groundwater level position, Aqh parameter in q(GWL)-relationship and Bqh parameter in q(GWL)-relationship}. I asked you that whether I am right to consider the runoff generated from the individual plots or all plots with the groundwater level of observation wells (7 shallow and 13 deep) in the study site to get those parameters [Aqh parameter in q(GWL)-relationship and Bqh parameter in q(GWL)-relationship] or not. According to you I will take reference as soil surface because our study site is almost flat but what will be the other relations/parameters. I hope you will make some clarification and needful suggestion to go forward. Let me wait for your reply.
Thanking you.

Laxmi Narayan Sethi
Dear all,
I am facing problems in assigning drainage boundary condition{The reference groundwater level position, Aqh parameter in q(GWL)-relationship and Bqh parameter in q(GWL)-relationship}. I hope you will make some clarification and needful suggestion to go forward. Thanking you.
Laxmi Narayan Sethi


Dear All,
Can You help me to know How to calibrate the Hydrus-2D model and also the validation in the field conditions. I have the data related to soil texture, bulk density, field capacity and wilting point etc.
Is it possible to simulate water flow in vertical and horizontal direction without calibration the HYDRUS 2D model?
Thank you.
Yours sincerely,
Laxmi Narayan Sethi


Dear Laxmi,
You can obviously use Hydrus without calibration. The problem may, however, be that the model predictions will not (very likely) correspond with your field observations. If that is the case, then some calibration may be necessary.

Hydrus can use a lot of various data information to be calibrated against. It can use measured pressure heads, water contents, actual and cumulative boundary fluxes, concentrations, and so on, to define the objective function, against which the model is calibrated. See examples in the "Inverse" workspace and the online help (where definition of the objective function can be found).

You can also look at the three chapters in the Soil Science Society of America book that reviews inverse approaches as used in vadose zone hydrology (below).

Jirka

99

Šimùnek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

Hopmans, J. W., J. Šimùnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Šimùnek, J., D. Jacques, J. W. Hopmans, M. Inoue, M. Flury, and M. Th. van Genuchten, Solute Transport During Variably-Saturated Flow - Inverse Methods, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 6.6, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 1435-1449, 2002.

http://envisci.ucr.edu/faculty/jsimunek/default.htm
Dear Jirka,
Thanks for your reply. Still I have question about the calibration. Though I have daily water content data, but I am not able to calibrate the HYDRUS-2D. Can you suggest me that where I will put the water content data in the HYDRUS 2D model. Is there any option for calibration. If yes then please help me to calibrate the model for my field condition.
One more question is about the initial boundary condition. Actually I am doing the experiment in the variable saturated domain of rice field where I have one barrier to retain 5/10 cm depth of water depending upon the intensity of rainfall. I am not adding any water for maintaining the depth of ponding. What boundary condition to be defined intially (constant pressure or variable pressure head).
Can you help me send the chapters of the book you mentioned for the calibration or the source of availability of book.
Thanking you. waiting for the reply soon.

Laxmi Narayan


You need to have the "inverse solution?" checked at main processes. Then you select observation nodes, and in "Inverse data" specify "time water_content Kode=2 Obs_node_Number Weight".

Hydrus does not calculate ponding on the surface due to excess rainfall. It assumes that all excess water is removed by surface runoff.

J.


Dear Jirka,
Thanks a lot for your reply. I have to quantify the lateral flow through the field boundary of trapezoidal shape and height 30 cm depending upon the ponding depth in the field due to excess rainfall. So, I am in diellema that what will be the boundary in that situation. Though the model assumes the excees rainfall as runoff from the field but is it possible to quantify the effect of excess rainfall on lateral and vertical flow

components? can you suggest about the limit (whether the satuarated moisture content of the each node) of the excess rainfall where runoff occurs.
Give me the aidea that what to do for my condition.

Thanking you. waiting for your reply.
Laxmi Narayan


The nodes that have atmospheric BC and pressure head equal to zero are the nodes where the irrigation (precipitation) rate exceeded the infiltration capacity. Therefore the flux BC was switched to head BC with zero pressure (in the Boundary.out file, you can see in which nodes this happened - Kode=4, as well as the part of the flux diverted to surface runoff). The total surface runoff is then equal to potential flux (irrigation) - infiltration (or actual flux).

Jirka
**ID = 86, Hydrus1D speed**

I have computer with 2.4GHz running Windows XP (1GB ram). When I installed Hydrus1D I noticed that calculations are as fast as on my notebook with 650Mhz and Windows 2000 (265MB ram). There is not a problem with writing on disk, since I use H1D_calc.exe, which writes on every 1000th time step. Is it possible, that Hydrus is not Windows XP optimized?


The HYDRUS that is in general release has full optimization as provided by the Microsoft Developer Studio. For this version there should be no decrease of speed between different operating systems.
Recently (last two years) we have been adding many new features to HYDRUS-1D and sharing that update with some of our friends and colleagues. That version does not have this full optimization on since it is not yet finalized. I have no idea how that version would get to you.

Jirka



**ID = 87, Alpha and n for specific soil textures**

I will be pursuing HYDRUS-1D and/or 2D. I have a really quick question that I have been trying to find an answer to for a few days. I know that you will be able to give quick and knowledgeable answer.

I have found many tables throughout various reports with alpha and n values for specific soil textures. Some vary greatly, but I keep finding reference to a table produced by Carsel and Parrish (1988). My question is:

Are VG parameters alpha and n specific to a certain soil type (texture), is there an acceptable range of values for each soil texture, or do they need to be calculated for each site with hydraulic variables?

Regards,
Spencer


Spencer:
Within the HYDRUS codes, and also separately, we provide Neural Network predictions of the VG parameters. They are based on the work by Schaap and colleagues, leading to the separate Rosetta software (http://www.ussl.ars.usda.gov/models/rosetta/rosetta.HTM) and Rosetta-Lite within HYDRUS. The software will let you predict the VG parameters from soil textural class, or from more detailed information (percent sand, silt, clay, bulk density, wilting point,...). We also include the Carsel table in HYDRUS to allow people to compare the Rosetta predictions with the earlier Carsel estimates. Hope this helps.

Rien van Genuchten


I am afraid that I have a bit of confusion regarding the alpha parameter in Rosetta. I am trying to perform a sensitivity to perturbing alpha paramters in my model. I have found a table that list the average value and one standard deviation (reported in parantheses) in Rosetta. The values are reported as log values; for example log -1.29 (0.65). If you were to take the inverse of those values, the result would be 0.051 and (4.46). Am to understand that the standard deviation reports 4.46 above and below 0.051?

I imagine there is probably an embarassingly simple answer to this; nonetheless, I am compelled to inquire. Thank you for your assistance. I have found both Hydrus and Rosetta to be outstanding programs.

Kindest Regards,
Joseph McCarthy


The standard deviation pertains to log(alpha. The confidence interval for log(alpha) hence runs from -1.29-0.65 to -1.29+0.65.


dear van Genuchten and all,
I have found some extra information in literature, but the question is, is this still valid or are the values different?

In the MvG equations the use of the n, m, l and the alpha were stated to be dimensionless parameters. They are used as fitting parameters in the closed form equations when m= 1-1/n (n>1). However I found in some literature the have the following physical meaning;

n = a pore size distribution index
alpha = the inverse of the air entry value (or bubbling pressure)
l = pore connectivity parameter

In the study of Brooks and Corey [lit 5.1] the value of l is assumed to be 2.0. In the study of Mualem {lit 5.2] the n is based on a statistical pore size distribution and the value of l is estimated as being 0.5 for many soils.

The equations are based on statistical data of soils and average values of soils. It has proven to be valid for many soils and accurate calculations can be made.

But my question is are the fitting parameters still connected to the physical properties as mentioned before or is this relation in non-excisting anymore ?
And if they still have a physical meaning can they be used to calculate pF curves (of non-soils) more accurate?

I know this questions brings us back to the basics but I am still interested in knowing

IJsbrand, others:
Yes, we tend to think that Rosetta is the best, but it sometimes enlightening to compare things with the Carsel and Parrish data to see the level of (dis)agreement.

The equations Brooks and Corey proposed, and our equations, are purely empirical functions in attempts to describe the water retention properties of soils (the same for most or all other equations; one may argue even for Kosugi's log-normal model). They are empirical, even though people have gone out of their way to assign physical significance to the parameters (n, alpha, hb, l, whatever). Defining alpha as 1/hb is such an attempt, which is formally incorrect: (1) many or most soils do not have a well-defined air entry value (that's why we went to that smooth function), and (2) alpha=1/hb is only an approximation (and a very poor for low n values). The same reasoning applied to l, which Mualem estimated in his equation to be 0.5 based on an analysis of some 40 soils (mostly coarse, repacked media). Marcel Schaap found that l on average is closer to -1.0, which does not make sense if l is interpreted as a pore-connectivity (or tortuosity) factor (physically, Se^l probably should decrease, rather than increase, when the water content decrease). Also, the B&C equations (for K) are based on a different pore-size distribution theory (Burdine), and the l=2 value should have no relationship with Mualem's l=0.5. Actually, our equations can be combined also with Burdine's theory (giving the approach Haverkamp and a few others prefer), while the B&C retention function can be combined with Mualem's equation. All those things are worked out in detail in the RETC manual.

One can take this point of empiricism even further with how WCR and WCS are viewed (residual and saturated water contents). For most dynamic soil water flow studies (infiltration especially), WCS is not porosity but some "saturated water content" (after Hillel) that is 10% to 30% less than porosity because of entrapped and dissolved air. Only for long-term saturated conditions (e.g., groundwater) will the effective WCS value approach porosity. All this is reason why I prefer to denote alpha, n and l (and WCR and WCS to the extrem) as empirical parameters taht define the shape of the hydraulic functions (empirical shape factors). Some purists among us may not like this.

Rien van G.

**ID = 88, Solute transport questions**

Several questions regarding solute transport in Hydrus 2D:
I've set selected nodes to Variable Flux BCs, and assigned them as pointer vectors "-2" for solute transport. When I go into the Time Variable BC editor I'm assigning the the vector "-2" concentrations in CValue2. Is this correct?

"Pulse Duration". If my total run time is 10,000 and I want to pulse 1 solute from time 2000 to time 5000, what is my pulse duration? 3000? 5000?

I'm not seeing any solute in my graphical display of results. I see "Concentration Solute 1" in the drop down window along with Water Content and Pressure, but no countours are visible when I run animation.

a) Correct. If you specify pointer=2 on a "variable flux BC", then you give concentration values in the column CValue2.
b) Pulse duration: This term has meening only for time independent boundary conditions. It will turn concentrations to zero on these boudnaries at time=Pulse duration. Time dependent boundary conditions are fully controlled in the "Time Variable Boundary Conditions" window.
c)Under "Options" click command "Color Spectrum for All Time Levels" and display specrum maps. Also check "Balance information" on whether there is any solute in the transport domain.

Jirka

In reviewing the output files (Hydrus 1d) for solute transport, I am trying to calculated strictly the cumulative mass of an organic solute arriving at the lower model boundary (in my case, the water table).
Looking in the solute output file, I see the Sum (cvbot) [M/L2] - how do I go about correcting the values to be expresses simpling in mass units? Do I need to calculate a corresponding volume, by which, to multiple my solute flux? I have reviewed T_LEVEL.out file, which peforms the water flux calculations. In there, I found the sum (vbot) [L], but the resulting units would leave me with [M/L]/

Any suggestions you have would be very helpful.
Joseph C. McCarthy

Joseph,
Solute transport units in HYDRUS-1D.

The actual solute flux is the product of solute concentration, c [M/L3], and water flux, q [L/T], thus c*q=[M/L2/T].

The cumulative solute flux is the integral over time of the actual solute flux, i.e., c*q*t=[M/L2]. You can interpret this as mass per unit surface area.

Jirka


## ID = 89, Flux across internal nodes

concerning water flow in field profiles I am looking for a comfortable way to calculate fluxes not only across boundaries but across internal nodes as well. Up to now I derive fluxes from the pressure head at observation points.
Benny.


Benny,
I have recently developed this option into the computational module. See the description below. It is going to be in the updated version. Hopefully in the fall of this year.

Jirka


Fluxes across internal lines (cross-sections).

HYDRUS-2D will be able calculate fluxes across the internal lines. This option is not supported at this moment by the graphical interface and therefore the user will have to provide an input file CrossSec.in himself into the "project folder". The input file crosssec.in has information about number of internal lines, which nodes define it, and which elements lay on one side of the line. Example is as follows:

nLines (Number of lines, Max 4)
1
nNodes(1....nLines) (number of nodes on the first, second, etc, line, max 250)
81
nElements(1....NLines) (number of elements upstream of the first line, max 500)
160
nn(1) (Nodes on particular lines. First line first. Each line starts at new line)
8 132 225 253 281 309 337 365 393 421
449 477 505 533 561 589 617 645 673 701
729 757 785 813 841 869 897 925 953 981
1009 1037 1065 1093 1121 1149 1177 1205 1233 1261
1289 1317 1345 1373 1401 1429 1457 1485 1513 1541
1569 1597 1625 1653 1681 1709 1737 1765 1793 1821
1849 1877 1905 1933 1961 1989 2017 2045 2073 2101
2129 2157 2185 2213 2241 2269 2297 2325 2353 2381
2409
ne(1) (Elements on one side of the line. Each line starts at new line)
15 16 73 74 131 132 189 190 247 248
305 306 363 364 421 422 479 480 537 538
595 596 653 654 711 712 769 770 827 828

885 886 943 944 1001 1002 1059 1060 1117 1118
1175 1176 1233 1234 1291 1292 1349 1350 1407 1408
1465 1466 1523 1524 1581 1582 1639 1640 1697 1698
1755 1756 1813 1814 1871 1872 1929 1930 1987 1988
2045 2046 2103 2104 2161 2162 2219 2220 2277 2278
2335 2336 2393 2394 2451 2452 2509 2510 2567 2568
2625 2626 2683 2684 2741 2742 2799 2800 2857 2858
2915 2916 2973 2974 3031 3032 3089 3090 3147 3148
3205 3206 3263 3264 3321 3322 3379 3380 3437 3438
3495 3496 3553 3554 3611 3612 3669 3670 3727 3728
3785 3786 3843 3844 3901 3902 3959 3960 4017 4018
4075 4076 4133 4134 4191 4192 4249 4250 4307 4308
4365 4366 4423 4424 4481 4482 4539 4540 4597 4598

The output is sent to the output file crosssec.out. It contains information at print times of fluxes in line nodes (sequentially as in crosssec.in), and at each time the actual and cumulative fluxes across each line. An example is as follows:

Fluxes across lines
Time Flux CumFlux
.001000 .2272E+03 .2272E+00
.001667 .1757E+03 .3443E+00
.002143 .1723E+03 .4264E+00
.002500 .1595E+03 .4833E+00
.002750 .1645E+03 .5244E+00
.002923 .1694E+03 .5537E+00
.003038 .1647E+03 .5728E+00

If there is more than one line, then first fluxes across all lines are printed and then cumulative fluxes.

## ID = 97, Floating point error

aztec_lc, 02/17/2003 : 04:23:00
I'm looking for an explanation of what a "floating point error" is. Also, where does Hydrus2D store the error log and how can I view it?
Thanks-
LC

Jirka, 02/17/2003 : 06:19:38
There is no error log generated by HYDRUS. That's a standard Windows response.
Floating point error may be caused by many things: nonconvergence, wrong input, ....

J.

**ID = 98, Free drainage boundary condition**

Findlater, 02/20/2003
Hi All,
I have a general question about this boundary condition. From what I understand, this condition imposes a unit gradient across the boundary, but where exactly does the water come from that is maintaining this gradient? Is it taken from within the system, or does this boundary in fact add water to the system?

Thanks for any input :)
laura@infomine.com

Jirka, 02/20/2003
Laura,
Free drainage can be specified only at the bottom of the transport domain. Since this BC imposes a unit gradient, then due to gravity there is a flux equal to the hydraulic conductivity for any particular pressure head at given time. Thus, this boundry condition can simulate only water outflow from the transport domain, and water comes from within the domain.

Jirka

**ID = 99, A-Level Output**

Hans, 02/24/2003
The A-Level Output is given at the times with variable boundary codition. I have serveral simulation problems with field data (therefore variable boundary coniditions!!), but my bottom boundary conidition is seepage face, nunfortunately this is not included in A-level output, especially if i have simulations runs over serveral years it is very adrious to get daily values for the seepage flux from the v_mean.out file. Is there any easy possibitly to adjust the A.level.out file so that also the seepage flux is included ??
many thanks for your answer

Jirka, 02/24/2003
Hans,
A_Level file gives information mainly about fluxes accross atmospehric BC, i.e., potential and actual infiltrations and transpirations. These info is given at times when BC is changed. Information about fluxes accross other BCs is printed into files v_mean.out and CumQ.out.

Jirka

**ID = 100, Windows 2000 XP…**

Jtourneb, 03/05/2003
Hello
Until today I use Hydrus2D under Window NT4.0 without any problem.
But we want to change our system into Window 2000, XP
We met a "severe" probleme : Target system OS not supported.
Do you know if it is possible and how to install Hydrus for this system ??? Can we just add new lines in file "OS" including Window 2000 or XP ?

Thanks
Julien

Mirek 03/05/2003
Hi Julien:
Hydrus-2D runs under Window 2000 and XP (I use Windows 2000 too). The problem that you describe was caused by a bug in the installation program (InstallShield scripts) and it was fixed about 2-3 years ago. Please download our latest version - it should work.
Regards Mirek

**ID = 101, Import initial conditions**

Preecha, 03/09/2003
Dear HYYDRUS users,
I am simulating a two-stage borehole test for field hydraulic conductivity.
I used general geometry for my probelm domain. I want to use water content profile from stage 1 for my initial conditions in stage 2. However, problem domian needs to be modified for greater depth of the borehole in stage 2. I could not import the water content profile from stage 1. Is there a way to do so?

Kind regards,
Preecha

Jirka 03/10/2003
Preecha,
I'm afraid there is not a simple way to do that. HYDRUS-2D has an option to import results from previous runs and use that as initial condition (pressure heads, water contents, temperatures, and/or concentrations) for subsequent simulation. But this option is only for the identical finite element mesh and transport domain.
More complicated option that would allow you to do that for different FE meshes is that you first import results for the original mesh, then modify the domain and mesh, and let hydrus interpolate the results. Interpolation will be done only for the original transport domain, and you will have to supply initial condition for the rest of the domain manualy.

Jirka

## ID = 102, Nodal drain flux calculation

pcastanheira 03/11/2003
I?m trying calculating the drain flux from a nodal drain flux with Vimoke adjustment. I can I do that? Thanks
Best Regards
Paulo Castanheira

Jirka, 03/11/2003
Paolo,
Read Section 5.3.7 at pages 51-52 of the HYDRUS-2D manual. You can get slightly more information from the SWMS-2D manual (also on the HYDRUS CD)at p. 27-31 and in related oroginal publications.
Jirka

Fipps, G., R. W. Skaggs, and J. L. Nieber. 1986. Drains as a boundary condition in finite elements, Water Resour. Res., 22(11), 1613?1621.

Rogers, J. S., J. L. Fouss. 1989. Hydraulic conductivity determination from vertical and horizontal drains in layered soil profiles, Transaction of the ASAE, 32(2), 589-595.

Vimoke, B. S., and G. S. Taylor. 1962. Simulating water flow in soil with an electric resistance network, Report No. 41-65, 51 p., Soil and Water Conserv. Res. Div., U. S. Agric. Res. Serv., Columbus, OH.

Vimoke, B. S., T. D. Tura, T. J. Thiel, and G. S. Taylor. 1963. Improvements in construction and use of resistance networks for studying drainage problems, Soil Sci. Soc. Am. Proc., 26(2), 203?207.

pcastanheira 03/11/2003
Thanks for your answer. I understand the Vimoke approach? but how can I calculate the nodal flux with Hydrus 2D?
Thanks

Paulo Castanheira

Jirka, 03/11/2003
In the boundary module you select the node that you want to represent the drain, then you enter requested parameters (Aqh, Bqh, and GWL0L).

Results: Drain flux is in the v_mean.out file in the column vDrain. Corresponding cumulative flux is in the Cumq.out file.

Some information is also printed to the boudnary.out file.
Jirka


**ID = 103, Radial coordinates**

Preecha, 03/12/2003
Dear HYDRUS users,
I wonder if HYDRUS can operate in radial coordinates (r and z).
I only saw the (x,y) coordinates in HYDRUS.
Kind regards,
Preecha

Mirek, 03/12/2003
Preecha:
Our current version (2.xx) supports only 2D Cartesian coordinate system (X,Y). Other coordinate systems (X,Y,Z-Cylindrical, Spherical, Polar and user defined local coordinate systems) will be supported in new version 3 (Hydrus-3D).

Regards Mirek


Jirka, 03/14/2003
Preecha,
The graphical interface of HYDRUS-2D always works with only two coordinates x and y. The computational module, however, can interpret them in different ways depending on information entered in the "geometry information" dialog window.

a) If you enter flow in a "horizontal plane" than the code interpret both x and y az horizontal coordinates, and hence ignores gravity.
b) If you enter "Vertical Plane", then the code interprets the y-coordinate as a vertical coordinate and x as horizontal.
c) If you enter "Axisymmetrical Vertical Flow" then y is interpreted as vertical coordinate and x as radial coordinate. Thus this option can handle axisymmetrical radial coordinates.

Jirka


**ID = 104, Saturated soil initial condition**

Pcastanheira, 03/12/2003
Dear HYDRUS users,
I am simulating a water flow to drain in agricultural subsurface drainage. However I have one issue witch I don?t understand.

My initial condition is: all soil saturated from the bottom to the top, so the pressure head is zero in entire domain? If I set this condition the result are non-senses. What I?m doing wrong?

Best regards
Paulo Castanheira


Jirka 03/13/2003
Paolo,
Well, you must be doing something wrong. In the Direct1 workspace there are two examples on drained profiles. I took one of them, the example "Drainage", changed the initial condition to fully saturated soil profile, and rerun it without any problems. Make sure that your initial condition is hydrostatic equilibrium in the saturated zone.

Jirka


**ID = 105, Anion exclusion effect**

Admin, 03/14/2003
I am interested in modelling the percolation of a sodium-chloride brine from a surface pit to a shallow water table through clay soils, and need to know something about the Hydrus-2D model.

Anion exclusion is the repulsion of the negatively charged ions (chloride, sulfate, bicarbonate) from the negatively charged clay particles which forces the anions into the macropores of the matrix. This phenomena results in incredibly fast travel times for some ions (like chloride) through the soils. Does the model take into account the anion exclusion effect? If so, how does it do it?

For more information on this effect, see:
* Thomas, G. W. and A.R. Swoboda. 1970. Anion exclusion effects on chloride movement in soil. Soil Sci. 110:163-166.
* McMahon, M.A. and G.W. Thomas. 1974. Chloride and tritiated water
flow in distrubed and undisturbed soil cores. Soil Sci. Soc. Amer. Proc. 38:727-732.

A prompt answer would be appreciated.
Mark S. Dalton


Jirka 03/14/2003
Mark,
There are two ways in which you can model anion exclusion using HYDRUS models.
1/ Both HYDRUS-1D and HYDRUS-2D implement mobile-immobile solute transport concept. Using this option you can specify which amount of the water content is excluded for solute transport. The pore velocities are then larger since Darcy flux is divided by mobile water content (instead of the total water content).

2/ In both codes there is no restriction on the value of the Kd adsorption coefficient. If Kd>0, the retardation coefficient is larger than 1 and solute is slowed down (retarded). If Kd<0, the retardation coefficient is smaller than 1 and solute is accelerated. One can use this option to model anion exclusion, which typically shows retardation coefficients slightly smaller than one.

Best regards
Jirka

rvang 03/14/2003
Yes, use a Kd<0. That is the classic way of doing things. Perhaps have a look also at CFITM, and especially CFITIM within STANMOD on our CD. CFITM runs without authorization. Example 3 deals with anion exclusion. Thanks.

Rien van Genuchten
George E. Brown, Jr. Salinity Laboratory, USDA-ARS

## ID = 106, Inverse solution

Sickfish, 03/21/2003
I have some questions about the parameter optimaization using inverse method. How many observation nodes are needed in a specific layer to yield a "healthy" solution? I have 3 to 4 layers in my model. Can I invert all parameters of these layers at the same time by defining observation nodes in these three layers and giving the observed water content at different time? Sometimes, I got an error message like: Floating point error! zero divide. Does that mean the Levenburg-Marquardt algorithm encountered a zero pivot error? What could be the reasons that cause it?

Jirka 03/21/2003
One observation node per layer should be enough. However, it it never a good idea to start optimizing parameters for all layers simultaneously. One should start slowly. One can assume first that the entire profile is homogeneous and only later start optimizing parameters of individual layers. Even then it is better to start from the first layer, and optimize parameters of one layer at a time. Only at the end one can try to finalize optimizations for parameters of multiple layers.

The Marquard Levenberg routine does not check whether used values are physically realistic. It is good to provide constraints on optimized parameters.

Jirka

## ID = 107, FE-mesh import/export and fortran sources

Admin 03/25/2003 :
From: kiyoshi yamada <ktaycici@cc.rim.or.jp>

1) I understand that this model is best used when HYDRUS-2D & MESHGEN-2D is used together. It seems difficult to export generated meshes or import meshes by other formats such as text format etc. My question is that it is possible to export/import(copy/paste etc.) meshes through HYDRUS-2D/MESHGEN-2D?.

2) Is HYDRUS-2D fortran source code available? Input format for HYDRUS-2D fortran executable available?

kiyoshi yamada
Jirka, 03/25/2003 : 13:35:51
Kiyoshi,

1/ MESHGEN-2D does not support importing finite element meshes generated or prepared by other software. It is posible to export meshes generated by MESHGEN-2D by saving it into an ASCII file using command "File->Save as Text File".

2/ We share FORTRAN source code for the h2d_calc.exe, i.e., computational module for direct problem, with users that obtained the full HYDRUS2D/MESGHEN2D software package from IGWMC on request.

Best regards
Jirka Simunek


## ID = 108, Cumulative drainage units

racefan318, 03/28/2003
I am trying to model a 90cm landfill cap to determine annual drainage through the cap. After running hydrus2d, the cumulative Free drainage seems to be higher than the total precipitation.
Precip = 24cm
Drain = 3200cc

I am looking at a 1cm horizontal, so it doesnot seem to be a factoring issue.

Any thoughts, or have I just not figured this stuff out yet?
Thanks
Brad


Jirka, 03/31/2003
Brad,
Obviously something is wrong. But from your description I cannot say what. Does the program converge or do you have nonconvergence most of the time? Look at our other examples so that you can see what temporal and spatial discretization we typically use to solve variably-saturated flow problems.

Regards,
Jirka

**ID = 109, Self learning**

Zal, 03/31/2003
Dear Moderator,
I would like learn how to run the hydrus2D model into several condition based on some example that come with the model. I need to know "the story" (field condition) behind the example i.e. the background of the field. Some of them are available on "the manual" or on "the tutorial" options. However, not all of them there.
If it is possible, could you send them to me please?
I really appreciate your help to guide my self learning.
Thank you.
Best regards,
Afrizal

Jirka, 03/31/2003
Afrizal,
Let me know in which examples you are interested in particular. I do not have any texts with examples beyond what is written in the manual. However, several of the examples were described in some journal articles. But some are purely hypothetical.

Also some examples are described in tutorials. Click tutorials above.
Best regards
Jirka

Zal 04/03/2003
Dear Jirka,
Thank You fro responding my email. Actually I want many of them, but at this time, I am interested to learn more about "Capbar", "Dam", "Dike", and "Fluxinf".
I would be glad if you can also tell me which article that describe some of those example.
Thank You for your help.
Best Regards,
Afrizal

Jirka 04/03/2003
Dear Afrizal,
The CapBar2 example is described in detail in:

Mallants, D., G. Volckaert, and J. Marivoet, Sensitivity of protective barrier performance to changes in rainfall rate, Waste Management 19, 467-475, 1999.

The other examples you mentioned (Dam, Dike, and Fluxinf) are purely hypothetical and thus I have no text that would describe them.

Regards
Jirka

## ID = 110, Landfill cap modelling

racefan318 03/31/2003
I am trying to model a 3' wood-ash landfill cap using Hydrus 2d. The values I need from the program are runoff, evaporation, transpiration, storage, and drainage through the layer; over a period of one year.

I have found the output of cumulative free drainage flux, root uptake flux, and atmospheric flux. Where can I find the storage, runoff and evaporation?
Brad

Jirka 03/31/2003
Brad,
HYDRUS takes the values of precipitation and evaporation, substract them and applies the result as the upper boundary flux. Then it uses only this value as potential flux, and calculates actual flux (reduction due to surface runoff for precipitation and due to dry surface for evaporation). HYDRUS does not print runoff values or evaporation by itself. You can, however, easily get them looking at the t_level.out file. If during the precipitation there is a reduction of the potential flux, then the difference is runoff. If there is an potential evaporation, and that is reduced, then you get actual evaporation.

Storage is printed to the balance.out file.
Jirka

## ID = 111, Boundary condition

stoefen@tuhh.de 04/01/2003
Hello everybody,
I'm trying to simulate a contamination in the unsaturated zone and the plume of that contamination in the saturated zone. The since I'm trying many combinations of length of the contamination and ground water flow velocity some of the solute might eventually reach the outflow end of the modelling domain. I'm wondering what boundary condition do I have to assign to the outflow boundary (where the groundwater flow will transport solute out of the modelling domain). I used a third type boundary condition with a concentration of 0, the outcome looks exactly the way I expected it to look like, but the mass balance error for the solute is continually rising during the simulation. In some cases it eventually reached 100%. Is this because of the boundary condition? Do you have any other suggestions what boundary condition to use in this case? There is the option of using a zero gradient boundary condition in HYDRUS 1d. Would a boundary condition like that be a better choice for this case?

Regards,

Heinke

Jirka 04/01/2003
Heinke,
In hydrus-2d, the Cauchy (third-type) boudnary condition is automatically switched into a Neumann boundary conditions at the output (see the discussion in the manual). Thus using the Cuachy BC at the outflow BC is correct.

Jirka

## ID = 112, Difference in mass balance

racefan318 04/01/2003
After running the program and looking at mass balance data from the A_Level.out file, it doesnt seem to add up.
Cum. Actual Atmospheric Flux = -347cc
Cum. Act. Free Drainage = 125cc
Cum. Act. Root Uptake = 67cc

What happened to the other 155cc ?

Brad Taylor, EI

Jirka 04/01/2003
Brad,
I would expect that the rest of water (155cc) is the change in storage. You can find that info in the mass balance output. There is also information about the numerical mass balance error, which is usually below 1%. If it is higher then your discretization of the problem is not adequate.

Jirka

## ID = 115, Upstream weighting

mizo316,  04/10/2003
Hi,
I would like to ask about the scheme used for the transport equation. Is it an upwind scheme?
Thanks.

Jirka 04/10/2003

The user can select both the time weighting scheme (explicit, Crank-Nicholson, implicit) and/or space wighting scheme (Galerkin finite elements (thus without upstream weighting), upstream weighting scheme, and Galerkin finite elements with artificial dispersion). The selection is done in the "Solute Transport - General Information" dialog window.

Jirka


## ID = 116, Hydraulic conductivity

mizo316 04/17/2003
Hi all,
When the model generates a hydraulic conductivity field using the stochastic distribution factors, is the average k of that field a geometric mean or an arithmetic mean?

Also, how can I make sure that I am not generating the same field twice or more, if I am trying to do something similar to Monte Carlo simulations, is there a seed number for each field generated that I can check?

Thanks.


Jirka 04/21/2003
Below is the program that we use to generate random fields.

Jirka

* Source file GENER.FOR ||||||||||||||||||||||||||||||||||||||||||||||||||

```
subroutine Gener [dllexport]
! (NumNP ,
! x ,
! y ,
! Bxz ,
c ! B ,
! E ,
! xLC ,
! yLC ,
! ZMean ,
! ZSTD ,
! iLogNorm)

* Two Dimensional Simulation of AutoCorrelated Random Variables
* From: Mejia, 1974, Water Resour. Res., 10:705-711'
* And: Freeze, 1980, Water Resour. Res., 16:391 (Has Some Errors)'

* SEED - Random number
* NTRM - Number of harminics (50< number <5000)
```

```
* A - exp. correlation (A) (0-not correlated)
* A=1./LC
* xLC - correlation length in respect of x
* yLC - correlation length in respect of y
* ZMEAN - Mean of Ln-transformed B
* ZSTD - STD of Ln-transformed B

implicit double precision (A-H,O-Z)
double precision E(NumNP),x(NumNP),y(NumNP)!,B(NumNP)
integer iLogNorm
double precision Gam(5000),Phi(5000),WM(5000)
real Bxz(NumNP),xLC,yLC,ZMean,ZSTD
data NTRM /2000/
* data A, ZMean, ZSTD /.25d0, 1.60, 1.25/

Pi2=4.0d0*datan(9.99999999999d20)
if(xLC.ne.0.) then
A=1.d0/xLC
Anis=yLC/xLC
if(Anis.lt.0.001) Anis=0.001
else
A=0.d0
Anis=1.d0
end if
call setseed(seedc)
seed=seedc

* Calculate random elements

if(A.ne.0.0d0) then
do 12 i=1,NTRM
11 GG=DRAND(Seed)
if(GG.gt.1.000d0) goto 11
Gam(i)=Pi2*DRAND(Seed)
Phi(i)=Pi2*DRAND(Seed)
WM(i)=A*dsqrt(1.0d0/((1.0d0-GG)*(1.0d0-GG))-1.0d0)
12 continue
end if

* Fof each s,y calculate synthetic variable

do 16 i=1,NumNP
E(i)=0.0d0
if(A.eq.0.0d0) then
E(i)=DRAND(Seed)
E(i)=RZ1F(E(i))
else
do 15 j=1,NTRM
T1=WM(j)*(x(i)*dcos(Gam(j))+y(i)/Anis*dsin(Gam(j)))+Phi(j)
if(T1.lt.100.0) goto 14
```

```
T2=float(jfix(T1/PI2))-1.0d0
T1=T1-Pi2*T2
14 E(i)=E(i)+dcos(T1)
15 continue
E(i)=dsqrt(2.0d0/NTRM)*E(i)
end if
if(iLogNorm.eq.0) then
Bxz(i)=ZMean+ZSTD*E(i)
else
c Bxz(i)=dexp(ZMean+ZSTD*E(i))
Bxz(i)=ZMean+ZSTD*E(i)
end if
16 continue


return
end


*********************************************************************
* FUNCTION RZ1F(X) -- MAKES A UNIFORM RANDOM VARIABLE INTO
* A NORMAL RANDOM VARIABLE OF MEAN 0.0 AND
* VARIANCE 1.0
*********************************************************************
*********************************************************************
* DRAND(X) -- RANDOM NUMBER GENERATOR (PORTABLE)
* FROM:
* LINUS SCHRAGE -- UNIV. OF CHICAGO
* ACM TRANSACTIONS ON MATHEMATICAL SOFTWARE
* VOL. 5, NO. 2, JUNE 1979, P 132-138.
* USES THE RECURSION: IX = IX*A MOD(P)
*********************************************************************


subroutine setseed(seedc)
real*4 seeda,seedb
real*8 seedc
integer*2 i100th
external random
external seed

seeda=20000
call gettim(ihours,mins,isecs,i100th)
call seed(i100th)
call random(seedb)
seedc=seeda*(1.+seedb)
return
end


*|||||||||||||||||||||||||||||||||||||||||||||||||||||||||||
```

**ID = 117, Location of input files**

3s 04/25/2003
hello all,
when i start to run the hydrus-2d model, it says that it can't find the selector.in file.
this is what the program says:
open file error in file:
D:\documents and Settings\antione\...\Selector

the file is in the specified directory. What could be the problem?


Jirka 04/25/2003
If the graphical interface can open the project, which means that the selector.in file must be there, and you get this message, I can think of only one explanation. The path to the project is too long ("D:\documents and Settings\antione\...\Selector.in") and your project is burried somewhere deep in your file tree. The FORTRAN code can handle the path only up to 80 characters long. Just move your HYDRUS project higher in the file tree, e.g., "c:\Hydrus2d\direct\..").

Jirka


3s 04/26/2003
hello jirka,
I'm dries Capelle and are doing my thesis on irrigation, i try to simulate the watercontent distribution in a vertical plane with
a furrow at the top of the vertikal plane.

I have another question, I read in one of the questions (mizo316 invalid point error) that if you use an constant head BC you must give the initial conditions in pressure head. I only have watercontent values. is there an easy way to convert these values into pressure head values?
and another I also get the flaothing point error, could the use of the watercontent be that error?

Where can i read the log file on what the program does? i mean where can i read what the programs does in the doslike window that appears?

thank for the answer on the first questions.


Jirka 04/27/2003
One can easily convert water contents to pressure heads using the retention function. If you want to specify initial condition in terms of the water content, then dirichlet BC is also given in water content. You can, however, specify boundary condition in terms of the pressure heads using the time-variable head conditions (even when constant with time).

There is no log file. That's an unrelated windows message.

Jirka


3s 04/28/2003
Sorry to disturb again,
I still have that "flaoting point error invalid number" problem
my question is, when i specify the initial conditions (this time in pressure head) Can Hydrus-2d work with a abrupt change in head?
i mean my field is a two-layered system with differentvalues of initial pressure head in that two layers, the interface is 1 node of width.

I only get output until 3/4 of the check.out, it stops before it reaches the end of the second material.

How can put in Seepage information?(this could be the error)
yours kindly
Dries
Jirka 04/28/2003
Dries,
There should not be the abrupt changes in pressure head in the soil profile. The pressure head is a continuous variable, i.e., variable that changes smoothly in space. This is contrary to other variables, such as water contents or conductivitites, that can have abrupt changes, such as at boundaries between two materials.

Before attempting to work on your own problems you should look at the examples distributed with HYDRUS and possibly also do tutorials, to understand how the software works. It is not easy to start using this software before prior knowledge about numerical modeling of nonlinear problems such as these.

Jirka


**ID = 118, Solute transport in tile drained field**

anjugaur 04/28/2003
I am trying to simulate chemical transport in a tile drained field. My first question is how to minimize run-time. It is taking forever to simulate a simple pulse problem. I did adjust time increment and intial time.

I applied input solute with constant flux and then in second case, I applied same solute rate via atmospheric (precipitaion=constant flux)boundary condition so that I could later apply variable boundary conditions. The resulting tile flux concentration pattern is coming out to be different under both cases while rate of input solute is same. May I know why?
Thanks. Anju


Jirka 04/28/2003
Anju,

Check the example test "Drainage" in the Direct1 workspace for an optimal set up for the tile drained field. In that example we simulated about 130 days and it took about half an hour. You can not expect to get results faster for such nonlinear problem.

You should certainly get the same solute fluxes out of the drain for cases when you apply either constant or atmospheric flux of the same value. Check whether you apply the same concentration. Whether you do not use the "Pulse duration" which applies only to constant fluxes (and thus not to atmospheric fluxes.

Jirka

## ID = 119, Effective diameter of tile drain

nasserga 04/30/2003
Dear Jirka
Would you please help me to know the method to calculate the effitive diameter of tile drain to simulate the water and solute transport using HYDRUS-2D.

Thank you
Dr. Gamal Khalil

Jirka 04/30/2003
Dear Gamal,
The effective drain diameter depends and can be calculated from the number and size of small openings in the drain tube [Mohammad and Skaggs, 1984. The effective diameter can be significantly different than the actual diameter, it can be up to 10 times smaller. Look up this reference and there should be some instruction on how to do that.

Mohammad, F. S., and R. W. Skaggs. 1983. Drain tube opening effects on drain inflow, J. Irrig. Drain. Div., Am. Soc. Civ. Eng., 109(4), 393-404.

Best regards
Jirka

## ID = 120, Spatial distribution of root uptake

Dear all,
I have received recently several questions dealing with the spatial distribution model for root water uptake in the BOUNDARY module of HYDRUS-2D (Options->Parameters for Root Distribution).

This root water uptake spatial distribution function has not been part of the initial release of the HYDRUS-2D. I have added it later when working with Jasper Vrugt and Jan Hopmans on the subject of models for spatial distribution of roots, and thought that others may also benefit from such option. That's why it is not

documented within the software. The function used in HYDRUS-2D is well described in two paper of Jasper Vrugt (see references below).

Best regards
Jirka

Vrugt, J. A., J. W. Hopmans, and J. Šimùnek, Calibration of a two-dimensional root water uptake model for a sprinkler-irrigated almond tree, Soil Sci. Soc. Am. J., 65(4), 1027-1037, 2001. See equation (6).

Vrugt, J.A., M. T. van Wijk, J. W. Hopmans, and J. Šimùnek, One-, two-, and three-dimensional root water uptake functions for transient modeling, Water Resour. Res., 37(10), 2457-2470, 2001.

Hi Jirka,
I have a couple of questions about the root water uptake distribution function added into HYDRUS based on the function that Vrugt, Hopmans and you defined in 2001 paper (SSSAJ). I am interested in using this function to define a root distribution for nectarine trees in an orchard. With the intention of understanding the function in HYDRUS better, I read the paper. I am a little confused after reading it.

When I compared the function beta (z) in HYDRUS book (Appendix III, Eqn. III.2) with the equation given in the paper (Eqn. 3), I noticed that the HYDRUS book carries a multiplier 1/lamda in two places. However, the paper explicitly says that lambda is eliminated from the original Raat's 1974 equation and new three parameters, zm, pz, and z* are introduced.

Since you said below that the function defined in the paper is implemented in HYDRUS-2D (from Options-> Parameters for root distribution window), I think that the functions would be identical. HYDRUS window where we defined the parameters is consistent with the info given in HYDRUS book. So, we need a lambda parameter in addition to zm, z*, and pz (or A as in HYDRUS). Why does HDYRUS have an extra parameter lambda? Does lambda have any physical meaning?

In HYDRUS book, there are plots of Beta(z) as a function of Pz and lambda? Are those chosen arbitrarily (0,2, 10 and 0.1,0.5, 1, respectively) or are there some ranges we can define? I don't have any information about the root distribution of nectarines, except max rooting depth. So, I don't know where to start. Any advice on this?

I also noticed that the paper talks about a root distribution for almond trees. So, I was hoping that I could use the values optimized in the paper (in Table 4) to start with. However, the values are given for zm, pz, and z*, not for lambda. I don't know what value to use for it.

One final thing..In Appendix III, it says that Eqn III 4 is obtained the same way as Eqn II.1 to define radial root dsitribution. Is that possible that it is actually Eqn III.2 which is referred to? Since the window requires delta parameter to be defined, I would assume that Eqn. III.4 is missing the delta parameter (a multiplier of 1/delta). If

this is the case, this equation is also inconsistent with the function defined in the paper since it does not have the gamma. Can you please help me clarify this confusion?


Thanks a lot,
Sevim


## ID = 121, Nonequilibrium sorption

Hi All,

I want to model kinetic leaching by using nonequilibrium sorption in HYDRUS2D. If i set fract=0, meaning none of the sites will have equilibrium sorption, do i still need to specify equilibrium sorption parameters (Kd, nu, beta) for this material? In other words, i am trying to figure out if it matters what i put in for equilibrium sorption parameter Kd, nu, and beta (since they will not be used) if i model that material with chemical nonequilibrium only (i.e. no chemical equilibrium).

Equation 3.9 suggests that the amount that will get sorbed will also depend on equilibrium sorption parameters. However, equation 3.16 suggests that the retardation will be unity.

I am having a difficult time figuring out first order rate sorption that is also always obeying the equilibrium distribution between the liquid and solid phase. Is this really what is happening if f is set to zero for a material? In other words, when kinetic sorption is occuring is it constrained by the equilibrium liquid solid distribution described by Kd, nu, and beta?

I would appreciate it if someone could clarify what i may be missing.
Thank you,
Defne


Defne,
No, it is not true. You still need to specify these parameters. HYDRUS uses the first-order kinetic equation (see equation 3.9 of the manual) for nonequilibrium sorption. The rate of sorption is proportional to this first order rate constant (omega in the manual and alpha in the graphical interface) and the disequilibrium, i.e., how far the system is from equilibrium. The equilibrium is defined with the adsorption isotherm for which you need these constants (i.e., Kd, nu, and beta).

Jirka


## ID = 123, Factorylink 6.6

parcbit 05/12/2003
We are two students from Belgium and we have to create a database with factoryLink 6.6. We are doing our final project in Spain.

We have problems and time is running.
One of the problem is when we install the cd of Factorylink 6.6,Monitor Pro Add-Ons.
The error is TARGET SYSTEM OS NOT SUPPORTED

If someone can tell us about it, it would help us a lot.


Mirek 05/12/2003
Hi,
This discussion forum is about Hydrus-2D problems, not about general IT problems.

However, although I have no experience with FactoryLink, I have experience with this error message (TARGET SYSTEM OS NOT SUPPORTED). It appears typically during installation of old programs under newer Windows (2000, XP,...) and usually it is an error in the InstallShield script. Solution: try to install it under older Windows (Win98, 95,...) or contact the program manufacturer.

Regards Mirek
**ID = 124, Balance output**

nzegre 05/13/2003
I am currently using HYDRUS 2-D to simulate sat/unsat through a cross section. I am interested in using the Balance output values per subregion for water flux coupled with nutrient concentration. I am confused as to use "inflow" or "volume" to calculate nutrient flux per time..thank you Nicolas Zegre


Jirka 05/13/2003
Nicolas,
a) "Volume" is simply the amount of water either in the entire transport domain (the first column) or in predefined subregions (the second and following columns).

b) "Inflow" is the change in the volume of water in the entire transport domain (the first column) or in predefined subregions (the second and following columns).

Jirka


Nzegre, 05/14/2003
Thank you for the reply. I am still a little confused..Let me explain. .

I have observed field data for nutrients and would like to calculate nutrient flux as a function of subsurface flow (time dependent). Many of my "inflow" values are negative (-)(subregion 1- surface soil horizon) to subregion 2 (impermeable subsoil). I am wondering if the (-) value in the subregion means that water was removed or gained from that subregion. In the Outputs section of the manual, for atmospheric flux, a positive (+) value means water was lost to the region, while in the balance.out, there is no notation of the meaning of the (-) or (+). I apologize for simple inquiries, the HYDRUS model is very new to me. Thank you

Jirka, 05/14/2003
Negative value of "Inflow" means that water is being removed from the domain or subregion at a given time. If you want to know change of water content between two given times, simply substract "Volume"s between those times.
J.


Nzegre, 05/16/2003
as per your comment above about subtracting the volume of (time +1 )from (time 0) to calculate the change of water content between two given times....should the change in volume of two given times always be equal to the inflow between those two times as well?

if the inflow and change in volume are different, should it be assumed results are not correct?thanks


Jirka, 05/16/2003
No, the change in "volume" between two times does not have to be the same as "inflow". The difference between two values of "Volume" reflects the change of water volume during the entire time intervals, while the "Inflow" is the actual change at a particular time. Thus one is an integral property (Volume2-Volume1), and one is the actual change (inflow).

Jirka


**ID = 125, Peclet number**

Dapul 05/13/2003
Hi,
1. For which node is the peclet number that comes in the run_inf.out file? As far as I understand, Peclet number can be calculated for each element: is the HYDRUS2D output representitive of the average peclet throughout the domain, or does it represent a particular element?

2. My peclet numbers come out really high for some part of my simulation (as high as 200, 300 when i have intense rain) even when I use the upstream weighing. I know i can't make the mesh finer than it is, is there another solution to this? Do my high peclet numbers suggest my results are really off because of numerical oscilations?

3. Finally, i would like to get a feel for how much of the contaminant movement (or alternatively what period during my simiulation) can be attributed to dispersion/diffusion versus advection. Is there an easy way to figure out this ratio for the entire system? (i.e. is it meaningful to calculate peclet for the entire system and if it is do i just need the velocity output file?)
Thanks,
Defne

Defne,
1. The Peclet and Courant numbers reported in the Run_inf.out output file are the largest element Peclet and Courant numbers over the entire grid for given time step. The code calculates these numbers for all elements, and report the largest ones.

2. Upstream weighting does not effect the Peclet number ($q*dx/theta/D$). Upstream weighting is a numerical technique that should prevent oscillations for transport problems with large Peclet numbers (like yours). Results could be OK even for large Peclet numbers if you use upstream weighting or introduce artificial dispersion (which decreases Peclet number by increasing dispersion). Look at the mass balances and results (for oscillations).

3. Peclet number is the ratio of the convective and dispersive flux. But it also reflect the density of the grid. So if your grid is irregular, it is difficult to use Peclet number to get this information.

Jirka
Dapul, 08/07/2003
Dear HYDRUSites,
I do have a uniform column that i am simulating.I am simulating 1D infiltration in an embankment. From what i understand in the literature, definition of the characteristic length for calculation of Pe is somewhat ambigous. For homogenous systems (which i can claim mine is) the characteristic length is the pore diameter. For column studies, people use the column length (and call it Brenner number). For simulations i have seen people use the depth of the simulated region.

What is your take on this matter? I want to use Pe to identify the importance of diffusion versus advection and i am undecided on whether one should use the simulated depth, finite element lentgh or the pore or particle radious.

Also, Jirka, you had mentioned HYDRUS2D reports the highest Pe number. Are the rest of the Pe values stored in any of the output files. (I don't need a change of code, just curious if the information is already there.)

Thank you,
Defne


Jirka 08/11/2003
Defne,
At each time step I calculate Peclet numbers for all elements and then report the largest in the Run_inf.out file. I do not store the other Peclet numbers, nore write them to any output file.

Jirka

## ID = 126, 1. converting to ASCII, 2. importing initial cons

Dapul 05/14/2003
Jirka thanks for your quick reply to my former question on Peclet.
I have two new questions:

1. I am running batch simulations in HYDRUS2D, keeping all file info under one name but moving the necessary output file to another folder, storing it, then rerunning hydrus2d for a different set of parameters. To calculate the flux in several different nodes, i convert conc1.out to an ASCII file then open it in excell, get the necessary concentrations and multiply them with the average velocity for that period which i get from v.out. I haven't figured out a way in visual basic to convert conc1.out to an ASCII file. Does someone know of a code that i can use to do this to avoid going into hydrus2d windows menu and hitting "convert conc1.out to ascii"? Alternatively, can i have conc1.out printed in ascii format?

2. What exactly is the process of importing initial concentrations from a previous run using the hydrus2d windows? I would like to be able to import concentrations without using the windows interface (using vbasic), any suggestions?

I realize my questions are more vbasic questions than HYDRUS2D but i will appreciate any help. I am using hydrus2d to run multiple simulations and i need to get around doing most of the things without using the windows interface.

Thank you,
Defne


Jirka 05/14/2003
Defne,
When importing the concentrations from the output file (binart file Concx.out) and using it as initial conditions the interface writes the concentration information into another binary input file (Domain.in). The code can alternatively use an ASCII file Domain.dat. You can easily find out the structure of the domain.dat file. Thus the only thing you need to know is the structure of the concx.out file. Below is the subroutine that writes this file at each time level. That should be sufficient for you to write visual basic code to solve your problem, i.e., reading concx.out and writing into domain.dat.

*****************************************************************

```
subroutine cOut(jS,NumNP,Conc,t,NSD,ierr,ii)

double precision t
dimension Conc(NSD,NumNP)

write(ii,err=901) sngl(t)
write(ii,err=901) (Conc(jS,j),j=1,NumNP)
return

* Error when writing into an output file
```

901 ierr=1
return
end


********************************************************************

Jirka


## ID = 127, Plane vs axisymetric

Nzegre, 05/15/2003
I am aware of the difference between horzintal/vertical plane flow and axisymetrix flow, but I am confused as to the units assigned to the plane flow (L2), instead of L3. Is the volume and inflow (L2) balance output simply the surface area of the volume/inflow?

I have simulated both vertical plane flow and axisymetric flow across a cross section of a watershed, and have received very different balance values. I tried to convert the vertical plan volume and inflows(L2) to L3 by multiplying by depth, but this did not work.

My question is: how can I convert a vertical plane volume (L2)balance output to L3? cheers, nicolas zegre


Jirka, 05/15/2003
Nicolas,
Two dimensional simulations are carried out in a two dimensional cross section of the transport domain. The area of the transport domain, as well as the volume of water in it, must therefore has L2 units. You can view these simulations as carried out in a domain that has one length [L] unit in the third direction, and view the units as [L3/L=L2]. Therefore you can get L3 units by multiplying results with the lenght in the third dimension.

Jirka


## ID = 128, Atmospheric BC

Dapul, 05/18/2003
Dear Jirka,
I have a question about how atmospheric data should be entered. I have one year's worth of quarter hourly precipitation data, and i was entering this data for every quarter hour resulting in 35040 pieces of data in the atmosphere.in. This appproach caueses hydrus2d to run much slower and takes up too much space in the atmosphere.in file causing my to run each year as a seperate simulation. So, i am looking for an alternative.

What is the best way to enter precipitation data for example if i have precipitation on days, 35, and 135 only and if i want to run the simulation for 1 year. I tried doing:

Day precipitation
35 0.3
135 0.3
365 0

but this does not work because the code assumes there is precipitation from day 35 all the way to day 365.

So i tried

Day precipitation
0 0
34 0
35 0.3
36 0
134 0
135 0.3
136 0
365 0

This input does work for the first precipitation event but not for the second. I get infiltration at day 35 and no infiltration until day 135. But after that, the precipitation doesn't stop and goes all the way to day 365. I think this approach will work for multiple rain events except the last event of theyear.

Can you help me figure out the best way to enter infiltration considering i have many precipitation events during a year?

Thank you,
Defne


Jirka, 05/19/2003
Defne,
If you enter atmospheric data for every 15 minutes, then HYDRUS needs to read this data as it runs and thus has to have a time interval of max of 15 minutes. That makes the run slow and one enters a lot of zeros.

You could enter that as follows:

34 0.0
35 0.3
134 0.0
135 0.3
365 0

This should be sufficient to provide all information needed, i.e., it rains only between 34-35 d and 134-135 d.

Jirka


## ID = 129, Infiltration

Nzegre, 05/22/2003
I have successfully run a simulation of subsurface flow across a cross section of a watershed. There is a loamy A-horizon (20cm) with a Bt-horizon (clay) below. as expected, theta is often greater in the Bt than the A-horizon.

However, when I compare simulated theta for A and B horizons to field observed theta (TDR), the simulated A-horizon theta is dramatically higher than observed. simulated Bt theta matches very well to field observed theta..

I have run multiple simulations (sensitivity analysis) with varying intial conditions (in water content)to account for such a high simulated A-horizon theta..

I am wondering if the high simualted theta for A-horzion could be a function of infiltration...Does the HYDRUS 2-D model solve for surface infiltration based on the surface soil horizon hydraulic properties?? or is it assumed all water infiltrates in the surface horizon ( except for a "no flow" BC, thus resulting in a higher simulated theta than observed?

I read/worked through example 1 (column infitration test...is "infiltration rate" or volume calculated and outputted in HYDRUS 2-D?? or does this need to be solved externally in an overland flow model?
I sincerely thank you!!!
nicolas


Jirka 05/22/2003
Nicolas,
That depends on what type of surface boundary condition you are using. If you use atmospheric boundary condition at the soil surface, then the code allows all water to infiltrate untill the infiltration capacity (ponding) is reached. Once ponding is reached the amount of water that infiltrates is equal to the infiltration capacity and all excessive water (prec-infil) is assumed to be removed by surface runoff. The infiltration capacity (and time to ponding) depends very strongly on the soil hydraulic properties of the surface layer.
Test1 assumes ponding at the soil surface and thus it actually calculates maximum infiltration capacity.

Jirka


## ID = 130, Error in iadmake

Mick, 05/29/2003
Dear Group

I'm new to hydrus and have constructed a simple 1/2 spacing simulation of a cross-section of a subsurface drainage problem with a no flow boundary condition on the sides and bottom of the domain and a constant pressure on the top. The drain is a seepage face, represented by a half circle on one side of my no flow boundary. One problem I have is that I refined the density of boundary points on the drain to get a better representation of it once I created a mesh, however when I do this and run the model I get an error "Error in Iadmake: Node 159 has > 15 Adjacencies" I've traced the problem to refining the boundary points on the drain (If I don't refine the boundary points then the simulation rans OK, but my drain is represented by a half hexagon - due to mesh generation!!!) and am wondering if someone can explain what the problem is exactly and what causes it.

Also for subsurface drainage problems in hydrus is it better to consider the groundwater divide (no flow boundary) at half drain spacing and directly intersecting the drain, which causes the drain in hydrus to be represented as a half circle on the edge of the domain or develop the problem with the drain in the centre of the domain and 1/2 spacing groundwater divides either side? What do most people do?

Regards Mick


Mirek 05/29/2003
Hi Mick,
Increase number of mesh points on the circle (Meshgen, View
-> Boundary_Points, Command "No of Points" on the left toolbar).

See also topic 52 in this forum:
http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=52

Mirek


Jirka 05/29/2003
Mick,
I would certainly define geometry with the drain on side (half circle) and middrain on the other side. Then you simulate only half of the transport domain than if you use the other option and save a lot of time.
Jirka


**ID = 131, Water content display**

mizo316 06/06/2003
Hi,
In the graphical display results, can we show Saturation percentages instead of water content values?
Thanks.

Jirka 06/06/2003
No.
J.

## ID = 132, Distribution of the phreatic surface as ascii-file

Hector, 06/16/2003
Hi there,
when comparing the effect of parameter variations on the distribution of the phreatic surface it would be advantageous to have the location of the psi=0 contour as ascii-files. Depending on the deviation of a parallel flow along a boundary, the pressure values at that particular boundary do not represent the location of the desired surface.
In Hydrus2D an algorithm for calculating the contours of a function $z = F(x,y)$ already exists, which yields the contours for the graphical display. It seems not to be too much of a struggle if there was the possibility to export an array containing the x/z-coordinate of the contour to a file. Is there any other way to obtain a file with the location of the phreatic (or any other particular isobar) contour?

Hector
Mirek, 06/16/2003 : 18:10:39
Hector:
The problem is not so simple (unfortunatelly). Contours in the graphical display of results are not drawn as continuous curves but as a set of independent line fragments (abscissas). There would have to be a special function for calculation and export of coordinates that you need. I think that this function could be very useful and we'll include it to our "task list" for Hydrus-3D. But if you really need such function very soon and if it could save a lot of your time then you can contact PC-Progress and try to order it.

May be Jirka will have a better idea how to help you but he is currently on the Hydrus course in Delft (so I'm not sure if he has time to read this forum right now).

Regards Mirek

## ID = 133, HYDRUS 1D transport problem

Meiwirth, 06/17/2003 : 14:59:31
Hello,
In a field experiment I applied pesticides to the soil surface and observed their transport through the unsaturated zone. I am now trying to simulate it with HYDRUS1D. I will first explain what I observed and then how I model it.

I applied the pesticides as an aerosol (100 ml/m2) of known concentration. Subsequent to the application, it didn't rain for 12 days and I did not observe any pesticides in the soil (the shallowest sampling depth is 10 cm). On the 13th day, it rained and the pesticides moved to considerable depth in less than a day.

In my model, the upper boundary condition is an atmospheric BC with surface layer and I impose rainfall and potential evapotranspiration at an hourly time step. For the solute transport, the boundary condition is a concentration flux BC. At the time of the pesticide application, I impose 0.1 mm of "rain" and the concentration of my aerosol (as cTop in the "Time variable boundary conditions" file). During 295 hours after the application there is no rain but intense evapotranspiration and thus an upward flux. Despite of this, the pesticides are immediately transported downwards and a pesticide peak at 30 cm depth appears only 2 days after the application… I have also tried using the concentration BC, but it didn't help.

How can the chemicals move downwards if there is no downwards flux? As I observed extremely rapid transport after the first rainfall, I use a high dispersivity (1000 mm in a 2500 mm long soil profile) and the mobile-immobile water concept with a high portion of immobile water (but even without immobile water the pesticides move downwards immediately).
Is this numerical dispersion? Can I do something to prevent it?

Thank you very much for your time.
Best regards,
Kirsten Meiwirth


Jirka, 06/20/2003 : 01:45:49
Kirsten,
It seems to me that you must have something wrong in the input data. If there is only 0.1 mm rain, and no additional flux, then there is no reason for the solute to move to the depth of 30 cm. Also it is certainly not due to numerical dispersion. Your dispersivity is so high that there is no reason for the numerical dispersion. But your actual physical dispersion could transport solute quite far. To me it seems unrealistic to have dispersivity of 100 cm. I would certainly decrease that value at least by one order.

You can send me your HYDRUS project files (JSimunek@ussl.ars.usda..gov) and I will have a brief look at it.

Jirka


**ID = 134, Contour lines**

mizo316, 06/20/2003 : 23:16:34
Hi all,
Can we show only one or two intervals of the water content saturation. I am interested in showing ONLY the line that dictate 90% stauration and over.
Thanks.


Jirka 06/20/2003 : 23:33:56
Mazon,

That's very simple. Under "Options->isoline parameters" select "offset" equal to your selected value, and then set "increment" to a large number. In this way the code will show only one contour line.

Jirka


## ID = 135, Scaling factors

Hi,
Generating a random field using scaling factors. When you try to print the generation, the distinction between colors is not easy. Can we show the generation as contour lines and can we select the scaling factors intervals as well.

Thanks.
Mazen


Mazen
See FAQ 13.
Jirka


I am wondering where does Hydrus get the mean value for scaling factors? I can only enter standard deviation values and correlation lengths. Does it use the value entered in the Soil Properties window?

Also, should the standard deviation entered be the natural log of the S.D.?
-M


Correct on both counts. J.


Once you have created the lognormally distributed field of Ks values. Does the spectrum used to represent the view have units of Ks or ln(Ks) values? It seems like they have to be ln(Ks)?
-M


Hi,
If you simply apply a constant scaling factor of K (lets say 5) to the whole domain, you'll see that the flux, which is directly proportional to k, will be multiplied by 5. Which signify that the scaling factor is a multiple of your mean k.
I believe that Jirka is on travel, and you can wait for him to reply correctly.
Thanks.

Is it possible to create multiple layers or zones of stochastic scaling factor? I have only been able to create stochastic fields for the entire model domain. I would like to describe and create unique stochastically defines parameter field for 3 layers.
-M


Hi,
At this moment it is possible to generate scaling factors only for the entire domain. It is not possible to split the domain and have different statistical properties of scaling factors for different subdomains. It is however, possible to have different mean values for different subdomains. That means that one can still have different material properties with different mean values for different parts of the domain, but the random field generated (over this deterministic field) is generated for the entire domain. I need to check how to do that, i.e, to have different stochastic properties for different parts of the domain. That's, however, not going to happen during summer (which I'm spending in Paris and Prague).

Wolfgang Durner found some errors in the way we generate stochastic fields. He helped us to update the entire part generating these fields (to generate both log normal and normal fields). I will try to include his changes in the update that we plan to post at the beginning of September.

Enjoy the summer! And do not do too much Hydrusing!
Jirka
Hi everybody,
I'm following Jirka's advice of not doing too much Hydrusing and read literature instead. Nevertheless, reading all this I have a question concerning the stochastic distribution of scaling factors.
How do I know the correlation length? And what s.d. value do I assume if I can't calculate it? Can I even apply stochasticly distributed scaling factors then?

thanks
Klaus


Klaus:
Good to see people having lots of fun Hydrusing during vacation time. Instead of fishing, why not indeed hydrus this summer. And yes, Jirka will show quickly severe withdrawal symptoms if he can't answer at least a few questions so once in a while. He is the champ. But let me try this one.

It seems like you need lots and lots of experimental data to calculate your correlation lengths for a particular application (e.g., a detailed lateral and vertical spatial variability study of water retention and conductivity functions). Very few good studies like this exists, and most only for the saturated conductivity (e.g., see Selker's book on Vadose Zone Processes). Nielsen et al. have done a few in Davis. So, many are purely theoretical exercises. Alternatively, of course, if no data are available for a specific site (which is normally the case), you may get some guidance from similar soils that my have been studied in the literature. The same for stochastically distributed scaling factors. Hope this helps.--Rien.

Hi All,
Assigning the standard deviation will dictate the range of variablity. But, shouldn't there be a limit for the standard deviation assigned in the case of generating scaling factors for water content?. Otherwise, the scaling factors generated will increase the water content beyound or below the saturantion and the residual water content respectively.

Thanks,
Mazen


See equation 2.26 of the manual.
Theta(h)=Theta_r+Alpha*(Theta*(h)-Theta*_r)

Theta*(h) - reference retention curve
Theta*_r - reference residual water content

The water content scaling factor (positive value) cannot scale the water content below Theta_r since it scales the water content difference. It can, however, scale the Theta_s value to be larger than 1 (imposible) and the user needs to check that.

Jirka
Hi Jirka,
I am interested in simulating a 2D infiltration problem using HYDRUS-2D and want to generate scaling factors. In your message posted earlier on the topic of scaling factors, you mentioned that there was a problem in the algorithm that is used to generate stochastic fields (including scaling factors). You also said that the algorithm will be revised and the changes will be implemented in the next release of HYDRUS-2D. I am currently running the version 2.008. Does this version have the changes already implemented in it or not?
Thanks,
Sevim Onsoy


The release of the latest update has been delayed since Eileen and Rien insisted that we have to first improve our security features. It could be out in a week or two.
Jirka


Hi Jirka and All,
Thanks for the reply. I will have another question related to scaling factors. Earlier in this topic, you mentioned that HYDRUS cannot be used to generate scaling factors for different regions in the domain. In my case, I will be working on a rectangular domain with 7 layers that have varying thicknesses in the horizontal direction (i.e., layer boundaries are not horizontal). I am interested in generating a different distribution of scaling factors for each layer. Since I cannot do this using HYDRUS, I will be editing the values of scaling factors in DOMAIN.dat. I have the scaling factors evaluated already at the locations of samples that we had soil hydraulic properties. So, I can the

data to evaluate variograms (and correlation lengths) and then generate scaling factors.

One difficulty I see now is that the geostatistical software (GSLIB) that I will be using to generate scaling factors assigns node numbers to a rectangular region differently. HYDRUS starts from the left lower corner, goes along the boundaries in counterclockwise, and then numbers the inner nodes from left to right. GSLIB starts the same way but continues to assign numbers from left to right. Here is an example how GSLIB and HYDRUS nodes are assigned for 4X4 domain. Note that the number stand for node numbers.

HYDRUS

10 9 8 7
11 15 16 6
12 13 14 5
1 2 3 4

GSLIB

13 14 15 16
9 10 11 12
5 6 7 8
1 2 3 4

I cannot seem to find an easy way to do transformation of node numbers between the two. Of course, it makes more sense to switch the node number in GSLIB output file in the same way HYDRUS has. If I get this part done, I will still need to figure out how to replace scaling factors in DOMAIN.dat for each layers. Since the layers are not horizontal, this step seems like a complicated one, too.

Any advice you can give me?
Thanks a lot,
Sevim


Sevim,
The reason why we number first the nodes at the boundary and then the internal nodes is due to the graphical interface. The graphical interface treats these nodes differently than internal nodes (such as displaying only boundary nodes, allowing users to specify boundary conditions, etc.). When we wrote the program we had to make a decision, what numbering is more convenient for us. And we chose the approach we use.

One can obviously find an algorithm to convert one set of numbering to another, since both have clear rules. I agree that it is not easy to do. But I had to do that for my internal structural generator. So it is doable but not simple.
Jirka

**ID = 136, hCritA value**

Dear sirs:
Would you please explain to me how to determine the hCritA value - Hydrus2D application - with an illustrative example if possible?

Thank you
Sofia Ghanimeh

Sophia,
Please, refer to the FAQ 6 fora definition of hCritA. I usually specify hCritA as being equal to about -150 m, although for fine textured material this value can be smaller(-500m),while for course-textured materials, such as sand,it can be smaller.

Th value of hCritA can be calculated from relative air humidity:

$$H\_r = exp(h*M*g/R/T)$$

where
H_r is relative air humidity (-)
h is the pressure head (hCritA) (m)
M is the molecular weight of water (0.018015 kg/mol)
g is the graviational acceleration (9.81 m/s2)
R is the universal gas constant (8.314 J/mol/K)
T is the temperature in Kelvins

Jirka

Dear Jirka
Thank you for answering my question. Yet I still need further clarification.
In your answer, you state first that you choose the value of hCritA according to the soil type (-150m or -500m). Then you give me an equation that is related only to climate conditions (Hr and Temperature)and totally independant of the soil type. I find a contradiction in the answer. Would you please explain.
Regards
Sofia Ghanimeh

Sophia,
There are two issues connected to hCritA. One is the actual value, which is a function of the atmospheric variables, and which is indeed independent of the soil type. The second issue is the numerical stability of the code. If you calculate hCritA from meteorological variables, you may get very low value in the order of hundreds of meters. However, in some soil, such as sands, there is not water for pressure heads already below about -10m. Then if you still keep the flux BC, you may encounter certain instabilities. Therefore I recommend to calculate hCritA from meteorological

variables, but check it against the retention curve, and if there is no (or small amount of) water for given pressure, then adjust it based on the retention curve.

Jirka


**ID = 137, Hydrus-1D, num. solut. not converged**

Hello,
when running Hydrus-1D (V. 2.01) with only water flow being simulated, I get the screen output "numerical solution has not converged". Although I still get results with the simulation I would like to know what this means.
Does it mean the result is not trustworthy?

According to the manual, I guess that my water content tolerance or pressure head tolerance are too small. (?)
So should I increase them until the numerical solution converges or are there other ways to solve the problem (e.g. my initial conditions)?

Is that the right forum to address such questions?
Where else can I get more information or help? (besides the manual and the help function in Hydrus)

Thanks a lot!
Roman


Roman,
If you get a screen output saing that "numerical solution has not converged", that's certainly a cause for concern. If you get it only once or twice, you still can have good results, but you should check the mass balance errors in the "Mass Balance Information". If this message appear more often then the simulation is not good. Your iteration criteria should be about 0.001 or less for water content and 1 cm or less for pressure                                                                                     head.

There is many reasons for this to happen. You may have unrealistic soil hydrauli parameters. You can have wrong temporal (1 s minimum allowed time step) or spatial (about 1 cm at the soil surface) distribution. Your boundary conditions may be unrealistic    (forcing    water    into    full    saturated    profile),    and    so    on.

Jirka


**ID = 138, Graphical results**

No records have been posted


**ID = 139, Temperature**

Hi guys,

I'm trying to find out about how much the temperature, which can be adjusted in the DOMAIN.IN file, matters.

At first I would like to know whether this really IS the water temperature and not any other temperature(I couln't find any information in the manual).

My second questions is on which functions of the HYDRUS-2D Program this temperature with the default value 20° has an influence. Does it also have an influence on the water flow simulations? I know that (in nature) there is an influence of water temperature on the hydraulic conductivity, so I'm interested in how far this temperature dependence can be simulated with HYDRUS.

I'm grateful for any answer,

Thanks in advance

Klaus


Klaus,

The actual temperature (initial value of heat flow is not calculated) does not affect anything unless it is requested. It can affect water flow through temperature dependence of viscosity and surface tension (see the manual). To get that you need to select "Temperature dependence" in the "Water flow parameters" dialog window. Solute transport and reactions parameters can also be made temperature dependent (again see the manual). To make them temperature dependent you need to select "Temperature dependence of transport and reaction parameters" in the "Solute Transport – General Information" dialog window.

I have been also recently working on coupled water and vapor flow, and heat transport. See the upcoming paper in WRR by Bridget Scanlon et al. (It may have been already published).

Jirka


Hello Jirka,

I have one more question concerning temperature:

I ran a few simulations at different temperatures (1°, 10°, 20°, 50°, 95°) in order to find out how big the influence of the temperature is on the simulation results. I ran two different kinds of simulation: one with time dependent precipitation and one without any precipitation.

For the first set of simulations I expected that warmer water would infiltrate more easily than cold water. The opposite was the case: The CumQA values of the 1° and 10° cold water were about 5% higher than the values for the warmer water.

For the second set of simulations I expected the warm water to drain more easily out of the domain, but again I was wrong: the cold water drained out more easily than the warm water.

For all simulations (of the first and second set), except the ones ran with the default temperature of 20°, the WatBalR value in the balance.out file was enourmously high though, with values up to 17 for the first set of simulations and up to 107 for the second set.

How can this be explained? Were my assumptions wrong?

Thanks in advance for the answer,

greetings

Klaus


Klaus,
I would not pay much attention to results you obtained since they will most probably be wrong. The typical mass balance error for HYDRUS simulations is on the order of 1% or much less (since we use mass conservative numerical scheme), and your simulations have huge errors. You must have something wrongly specified. I can't really say what, just some hints: Water content tolerance <0.001, Pressure head tolerance < 1 cm, initial and minimimal time steps about 1 s, and so on. Check the given examples for discretization, tolerances, and mass balance errors. Until you get mass balance errors below 1%, do not try to interpret the results, since they may not be meaningful.

Jirka


Hello everybody,
I have run a whole bunch of temperature dependent simulations now, with temperatures ranging between 10° and 20° Celsius.
When I keep the default temperature (20°) it works for about 70% of all simulations. But when I change the temperature, I almost allways get a big mass balance error, which makes the simulation results quite useless. I also tried different levels of discretisation, Water content tolerance, Pressure head tolerance, initial and minimimal time steps about, different soils - I still couldn't get rid of the mass balance error.
Has anybody else tried to run temperature dependent mass transport simulations? Any experiences to share?
Thanks in advance for any hints,
Klaus


Klaus,
I have done several runs with temperature dependence and I did not see any mass balance problems. I have sent you directly to your email address a zipped file that contains several projects and the code I'm using here. I do not remember fixing any errors concerning with temperature dependence, but it is possible.

0Test1 is the original Test1 project from the workspace Direct. Mass balance error= 0.011

0Test1a – the same project, except that I selected temperature dependence. Error=0.013

0Test1b – the same as 0Test1a, except that I simulate temperature as well, with initial temperature=20C and temperature of infiltrating water=30C. Error = 0.009

Jirka

**ID = 140, Diffusion in two layers**

Dear All,
More recently I am working on a diffusion problem where i am trying to model diffusion from a contaminated sand to a reactive uncontaminated material. I have very different Kds for each material (1 and 500 m3/kg) and one side is very contaminated (10000mg/kg) and the other very clean. When i run the simulation i get very high mass balance errors on the contaminant. I made my mesh much smaller (0.1mm)around the region where the two materials meet and it did not help a big deal. I tried extending the contaminated region to the clean reactive material and that was moderately helpful but not as good as i wanted it (it reduced relative mass error from 97 to 14%). I haven't tried minimizing the time step but i doubt that will have any affect since the simulations run very fast. I have two questions:

1. Is there another way to minimize the mass balance error when there are two distinct layers in contact? Hector had mentioned assigning properties to elements instead of nodes in a former post (3/16/2002). Jirka, may I have a copy of this code? I will probably also need it for some other work on pavement water movement modeling.

2. Is there a way to assign different diffusion coefficients to different materials? I would need this option because my materials have different tortuosities and i am including tortuosity in the diffusion coefficient.

Thank you,
Defne


Defne,
In addition to the mass balance error, look also at the absolute error and the total solute mass in the transport domain. It may be possible that the mass balance error is actually not bad at all. I calculate the relative mass balance error as a percentage of solute fluxes that flow across boundaries of the transport domain. In your case, you have no flux across boundaries and thus the absolute error is related to very small number (close to zero) and that may produce large relative mass balance errors. See also the manual on the way these numbers are calculated.

1. Assigning of properties to elements instead of nodes is implemented only for water flow, where the effects may be significant. I did not see reason for doing it also for solute transport, where I think the effects would be negligible.
2. HYDRUS reads in only one diffusion coefficient for each solute, and thus assumes that it is the same in all materials (which is correct assumption). It would need to be modified so that it can read diffusion coefficients not only for each solute, but also for each material. I can do that quite easily. Let me know if you really need it.

Jirka


Jirka,

You are right, my absolute mass balance errors are not bad at all. I am happy with the absolute errors and will use those as a basis for the success of this particular simulation.
Thank you,
Defne


**ID = 141, Simulating solid solute leaching**

thabo23 07/23/2003
Dear Jirka,
I trying to use HYDRUS-2D to simulate the leaching of a nitrate solid fertiliser. I want to use a furrow irrigation system and a solid fertiliser is placed on the side of the ridge at a particular depth. How do I get the 3rd boundary condition for this? Is there any best way that you can suggest?

How do I define the boundary condition (variable head) on the furrow if I want to see the leaching with time (especially the first half of the irrigation event). What I mean here is that once we start pumping water into the furrows the water starts building up with time. If I give a changing head in the variable time BC it can do that but then how does the model see the different heads in different nodes in the furrow. The node situated at the surface of the ponded water will have 0head and the node below that will have 1cm and another node below that will have 3 cm. How do I model this?

Thanks
Thabo


Jirka 07/23/2003
Thabo,
Fertilizer: It seems to me that you should place nitrate into the soil profile as an initial condition to given position. You can enter it either in the solution (solute concentration, which means that it is already dissolved), or you can trick the code and use the kinetic sorption-desorption option, and place it into a kinetically sorbed concentration, from which it will be slowly released through mass transfer.

Boundary conditions: Standard code automatically distributes variable pressure heads as hydrostatic pressures, i.e., it assigns specified head to the lowest node and then takes z-difference away from this pressure from nodes above. In the past I have actually been working on rising and falling levels of water in furrow and develop a special option that can deal exactly with that. It then specifies atmospheric BC in nodes above water level (while standard version would fix there negative pressures). If you are interested in this version send me email directly.

Jirka


valliyappant 08/16/2005
I read a couple of other topics in the hydrus2d forum (example: pesticide simulation - topic ID: 250) which are related to solute de-sorption from solid phase. Based on the

replies I tried simulating solute de-sorption from solid phase using Hydrus 2.05 version. I have problems with simulating solute de-sorption from the solid phase.

How do I use the concept of two-site sorption to obtain solute de-sorption from the solid phase?

1. Is it feasible to have initial solute concentration in the solid phase?
2. Is it feasible to alter the value of Kd in equation 3.9 (hydrus2d manual - chapter 3)to be negative? (OR)
3. Is it feasible to alter the value of w (or alpha) in equation 3.9 to be negative? (OR)
4. Is there some other way?

Thanks for your time and help.
waiting for your reply.


rvang 08/16/2005
Valliyappant:
The solute will desorb from the solid phase only when you leach with a solution that has a lower concentration. Either with the equilibrium or two-site sorption model. Your specific questions:

a. The initial sorbed concentraiton is determined by your initial concentration and the sorption parameters, notably Kd (see Solute transport - Reaction parameters)
b. Yes, specify Kd in "Solute Transport - Reaction Parameters", first entry. A negative value means anion exclusion (e.g., see CFITIM manual). You need to have a positive value if the solute is subject to sorption (even if it will desorb when leaching the soil with a lower concentration solution).
c. Yes, you can change w or alpha to whatever you want, but between 0 (no exchange takes place between solid and liquid phases) and infinity (it exchanges instantaneously, wich is essentially the same as using the equalibrium model). What is the purpose of using a negative w? You then force solute to adsorb/desorb against the concentration difference between liquid and solid phase (i.e., your force solute to desorb from a solid phase that has no solute anymore!); the program probably will crash, or do something weird.
d. Yes, specify an initial concentration and a Kd, and leach with a solute-free solution (or a solution with a lower concentration). Like leaching an initially contaminated soil profile.nse.

Hope my comments make sense.--Rien van G.


## ID = 142, Calibration

Hi All and Admins,
I would like to use hydrus 2d in my research project, but still confuse with term KALIBRATION and INVERSE MODELLING.
Do we need to calibrate hydrus 2d before using it?
Is that good enough just using Iverse modelling method to check wether Hydrus can predict the field condition corectly, instead of using calibration methods?

Thank you for the answer.
Best regards,
Afrizal


Dear Afrizal:
Except for some of the inverse theoreticians among us, calibration and inverse modeling are essentially the same in that a number of model parameters are fitted to observed data to get a good match of the model with available data (laboratory and/or field data). However, there is no need as such to calibrate HYDRUS-2D before using it. It all depends on the amount and quality of your data. In some cases your data may not be good enough to calibrate the model, or you may even have no data at all. In other cases you may have to rely on other means to get information about the problem (e.g., soil survey data and using pedotransfer functions to estimate the soil hydraulic properties, or using some pesticide database to get sorption and degradation parameters) in order to make predictions.

Hope this helps. Rien van Genuchten


## ID = 143, Observation point

soja0496, 08/06/2003
Hello Everyone,
As a beginner using hydrus 1d, I have a very elemetary question -perhaps. What is the simplest way to precisely insect an observation point?.The second question I have is what is the practical meaning of constant pressure head boundary condition?
thank you.
yours
Joseph(MSC student)


Jirka, 08/06/2003
Hi,
1) Click on insert, move mouse to desired position and click again. That should be it.
2) Constant pressure head:
a) ponding infiltration (with positive pressure)
b) tension infiltration (with negative pressure)
c) boundary with staning water, i.e., stream, well, etc.
d) Many others

J.


## ID = 144, Solute transport upper boundary condition

Hallo,
I'm trying to simulate a Bromide transport experiment in HYDRUS-1D. The mass of Bromide applied was 30,15 g/m² or 0,0377 mmol/cm². It was applied in a solution of 0,737 mol/L, therefore 0,512 L were applied.

I used "Concentration BC" as Upper Boundary Condition and set cTop = 0,0377 (mass units = mmol) and precip. = 0,512 (lenght units = cm) in the time variable boundary conditions.

a) Is this the correct way to describe the bromide application?

The Problem is: I observe a lower Bromide leaching than expected in the simulation. The cumulative leaching over the whole experiment yields only about 14 g/m², which is not even half of the applied Bromide, but it was expected that much more Bromide would leach.
b) Why could that be?

[The experiment ran over two years with over 1600 mm precipitation and a leaching of over 600 mm (that is more than the pore volume of the column)in the Hydrus Simulation. After about 300 days the concentration profiles show, that there is no bromide left in the profile and the cumulative Bromide outflow becomes stationary.]

c) My time step is in days, therefore the concentration boundary condition applies for the whole first day. Luckily on that day there was no other precipitation in the experiment than the bromide solution applied. If there was natural rainfall, I could not use that on the first day of the Simulation since that would increase the applied bromide mass. Right?

Thank you a lot for any help!
Roman


Roman,
Use the "Concentration (solute) flux BC". Then you control the volume of solute entering the transport domain. The amount of infiltrated solute is equal to Concentration*flux*time, i.e., $[M/L3]*[L/T]*[T]=[M/L2]$, e.g. kg/m2, g/cm2. It is important to use compatible units [L] in concentrations and the rest of hydrus.

You can check the cumulative fluxes at the top and bottom of the domain (solutex.out), with the amount of solute in the domain (balance.out). Adding these together should give you the error in balance.out.

J.


Hello Jirka,
thanks a lot for your help.
After correcting my earlier mistake, I still had an error of 35% (CncBalR in balance out). The cumulative fluxes are 0.0137 at top and 0.0286 at the bottom, adding to an error of 0.0149 (CncBalT) (amount in domain is negligible, bottom boundary cond. is free drainage).

I wanted to reduce this mass balance error. With a smaller minimum time step (now 0.00001 days) I could reduce CncBalR to 5%. I could accept this, although <1 % would be nice, but I wonder if my Courant number is too high may

**ID = 145, File too big...................**

Hello Jirka,

I am very grateful for your answers and of course the time spent. I however have another question which is, if one gets the warning or alet message" File is too big to be displaced entirely automatic selection has been made" does that mean the output results cannot be relied upon?

Well, explanation is welcomed from any member.
thank you.
Joseph.

Hello Joseph,
The chart has some limits given by the external component that we use to display results. Speaking generally: max. number of curves <=20 and max number of points on one curve <= 6000. But these criteria are actually more complex and depend on the graph that you want to display. Points or curves that overflow these limits are ignored and are not displayed.

Reagards
Mirek

hi Mirek,
Does this mean that the end of the file is ignored? So start at the beginning untill 6000 is reached and than stop? or...
I thought that it would take some points (max 6000) of the whole file to be displayed. I mean from the start to the end of the file.
Is this not true?
please tell me as I sometimes use the curves to back my story up.

IJsbrand,
For backing up your story, in case you haven't tried it yet, you can also plot your own graph in excel in which case you would not have to worry about max number of points. The plots in HYDRUS2D are all based on the output files which i commonly import to excel for plotting and manipulating as needed.
Just my two cents..
defne

Hi Dephne and all others,

Yes you are right .. Although am a bit worried with the manipulation of data...... Just joking ... No the other (a bit hidden)question that was more important; on what does the graphs in Hydrus-2d show actually? I would choose to see the overall picture (trends)rather than only the beginning of the results as often the final stage is the most interesting. Don't you agree? But is is handy to know what is actually displayed.
Furthermore greeting from an exceptional hot Holland (34 degrees C)
IJsbrand de Haan


Hello IJsbrand,
I've checked our code once more and unfortunately there is no "smart selection". The chart really displays only first 6000 points (or less) and the rest is not displayed. But in this case users are informed by the warning message ("File is too big..."). I remember we intended to use something better (a selection that you expected) but this has never been really done. This problem will be fixed in a new version (Hydrus2D/3D). If you want to display all points/curves, please export data for example to Excel and create charts there as Defne suggests...

Regards
Mirek


IJsbrand, I was wrong!!!
Yesterday I talked to Jirka about this problem and he said that he had created another function that really makes a "smart selection". I didn't know about it so I had not checked what function is really called (I checked only the original function). Sorry for the confusion. I hope that Jirka will explain how the smart selection is implemented as soon as he is back from holiday.

Regards
Mirek


Well, it is not that smart, only the best way I could come up with. When the maximum number of points that can be displayed is reached, then I reread the same file, but by reading every other value. If even then the max number is reached, the I read every third value, and so on. Thus the values are reliable, except that a regular selection is made. In all cases, the time interval covered should be the entire simulation time.

Jirka


Dear Mirek and Jirka,
I also have similar problem, but in my case the error message not file is too big. It says "hydrus has generate an error".
I was try to develope a domain of drainage (mesh generator), and suddenly hydrus closed automatically.
Can you tell me what happening?
Thank You.
Zal

Hi Zal,
did it happen in the Meshgen module during FE-mesh generation (the button "Make Mesh") or when exactly? If it was during mesh generation could you send me the project (all files in the project directory)? Please use WinZIP if you can. My e-mail address: mireks@pc-progress.cz

If the problem is in Meshgen then it would be good to create a new topic in this discussion forum (for example "Mesh generation error") because it doesn't concern this topic (file too big...)

Regards
Mirek


## ID = 146, Salt accumulation

Hello,
I'm now trying to use Hydrus-2D to simulate the salt accumulation with ion-exchange, but I can not get ideal result.

The condition is;
A 30cm long soil column is first saturated with a solution, which contains calcium ion and sodium ion(the concentration is 250 ppm respectively).
The evaporative flux is supplied on the upper boundary.
Soil type is loam and final time is 288h.

I have the experimental data, which is the ratio of calcium ion to sodium ion at the surface of the column changed as time progress (the ratio of calcium ion had decreased and sodium ion had increased, probably because of ion-exchange).

I want to represent this phenomenon in HYDRUS-2D.
I tried to simulate this, but the ratio of calcium ion to sodium ion was 1 to 1 at final time. Is it possible to represent this in HYDRUS-2D?
Thanks
Yoko


Hi Yoko,
I have never done an ion-exchange problem so i am not sure how one should set it up in HYDRUS2D. What exactly is going on in your column? What ions are being exchanged with what? I can't picture how one would model "the exchange" but i think you can try each ion to be retarded at different rates if that physically makes sense to you. For that, you would have to decide what isotherm values you need to assign for each ion. Your 1 to 1 ratio suggests you either did not assign any distribution coefficients or assigned the same value for both ions.
To actually make it "exchange" ions you might have to come up with a different way to set it up in HYDRUS2D which i am not sure how to do.
Defne

Yoko,
I'm on vacations in Prague and thus do not have much Hydrus stuff here with me. Anyway, I remember that there is one example on cation exchange that is described in both HYDRUS-1D and 2D manuals. Loot at the test examples. There is one that deals with cation exchange between calcium and magnesium, treated as nonlinear sorption using Langmuir isotherm.

Jirka

## ID = 147, Invalid cell reference!

Hi,
I am a new user of HYDRUS2D program. I am having difficulties with entering geometry information for a regtangular discretization of a 2D flow problem (vertical plane). I am actually editing the geometry information of an existing project to expand the horizontal dimension and to refine the mesh both in x and z directions. When I enter the new info in Geometry and Mesh Editor window (horizontal and vertical dimensions and # of horizontal and vertical colums), I get an error message stating "Invalid cell reference" before I proceed to the next window "Space discretization". This window does not carry any discretization information from the previous step. Does this indicate that the geometry information should be entered manually? Also, when I click on "previous" button to go back to Geometry and Mesh Editor window, HYDRUS2D generates an error and shuts down automatically.

I am wondering what is causing these problems to be generated and then causing the program to crash. Is there a limit on the number of nodes that can be specified? If so, maybe I am exceeding the limit.
Sevim
Sevim,
This message is indeed caused by exceeding the number of nodes allowed in the vertical direction. There is practically no limit in the vertical direction. I do not remember what that maximum number is (I do not have access to the source code at this moment). The limit is given by the grid module that does allow only limited number of columns.

Jirka

Dear Sevim
I've read in a HYDRUS-2D modeling book (Rassam, Simunek and van Genuchten) that the maximum allowed number of vertical and horizontal columns is 10,000. Hope this would help
Sofia

Sevim,

Sofia is correct. The maximum allowed number of nodes both in the horizontal and vertical direction, when using the internal mesh generator (simplified geometry), is 10000.

In the earlier versions of HYDRUS, due to the limits in the grid OCX (input spreadsheet), the max number of nodes in the horizontal direction was 250. After realizing that this may be limiting to some of our users, we have reprogramed the interface so that if the number of nodes is larger than 250, then the coordinates can be entered in vertical column (instead of in a row), thus avoiding this OCX limit.
Jirka
Dear Jirka and Sofia,
Your further information on this matter made me realize that I was using an older version of Hydrus2D since I was not able to enter 300 nodes in the x dirn, and the geometry info in the x dirn was given horizontally, not vertically as Jirka says it should be. I just downloaded the new version 2.008, and tested with the max allowed # of nodes. No problem as I expected.

Sevim


## ID = 148, Geomembrane modeling

Dear all:
I'm modeling water infiltration through landfill covers, which may include a geomembrane or a geotextile liner. Do you have previous experience or a suggestion on how to simulate (using Hydrus-2D) the effect of those liners on water movement?
Regards
Sofia




Hi Sofia,
Are you doing 1D or 2D? What is the information you have about your system. It sounds like you need to model your geomembrane as an impervious layer but i don't really know what its unsaturated parameters may be like. I imagine there should be some published data on it.
Defne


dear sofia,
I know that the name for geotextile most often is used for a membrane or textile that lets water pass uninterupted. By landfill covers they are often not used but covers that are (totally)impermeable. They do not transport water or so little that it can not be included
(I thought it was in the scale of; kd $1*10^{-18}$ ). The figure I am not confidence about but they are rigoursly written down in the demands for landfill covers so very easily found. In hydrus it is easilier to simulate them as an impermeable boundary. I did some simulations with roads which contaied such a cover. I used a very thin internal hole (double line) in the internal embankment with impermeable boundaries. But this

creates a simulation with a flow over the impermeable layer, so you can simulate also only the part underneath (and left and right)of the layer.

IJsbrand de Haan

Dear Ijsbrand and Defne
I'm working in 2D vertical plane, simulating water infiltration through a layered earthen landfill cover. I can't model the geomembrane as an impervious layer because I won't have any water infiltration through the landfill cover. Although this may be theoretically justified, it is practically wrong. All geomembranes start leaking few years after installation. Therefore I need to come up with certain hydraulic conductivity and unsaturated parameters that may give results close to reality.
One option may be to create holes in an impervious layer, yet the challenge is how many of them? And how big?

Concerning the geotextile issue, I'm not confusing it with geomembrane. I meant the GCL sheets, i.e. bentonite sheets sandwiched between two layers of geotextiles (to improve tensile strength) and sometimes adhered to a geomembrane.

Again, I'm readdressing this question to everybody: if you have a previous experience or a suggestion, I'll appreciate your help.

Sophia Ghanimeh

Hi sofia,
Sorry I just wanted to make clear that we are talking about the same geomembrane. Furthermore, If you want to model the behaviour of the geomembrane very well you need to know how it is leaking after a few years.

Maybe a option is to use a thin bentonite layer with preferential paths in the material. (Randomly added in Hydrus 2-d). This however creates a large gradient on a small distance whitch makes simulations difficult.

IJsbrand

Hi Sofia,
we used to make some simulations for capillary barrier cover systems and we found out, that the parameter you are looking for, the effective hydraulic conductivity of the liner seems to be very uncertain. The vendors "classificate" those as impermeable and nobody could give any estimation on the effective permeability of such liners under filed conditions. Without that information however model results remain speculative, at least in a predictive sense.

In Germany a series of large scale test-fields were installed in order to estimate the sealing performance of several cover systems (google around for Stefan Melchior and/or Markus Kämpf for further hints), however I do not know, if the effective conductivity of PE-liners was addressed in those field studies.

The only thing you could perform would be a kind of scenario study, in which you vary the conductivity (or the number of leaks per area) of such a liner.

Doing so you should consider the thickness of the geomembrane being only some mm. As you know in hydrus2d the conductivity is assigned node-wise, so any conductivity contrast would be "smeared" at the layer's interfaces. You should refine your discretization sufficiently in order to have a reasonable representation of the flow processes above and below the membrane.

Hector

## ID = 149, Lysimeter modelling, lower boundary condition

Hi!
I am trying to model the movement of water in a lysimeter study in relation to drip irrigation. At the bottom of the lysimeter (outside my domain) we have a 20 cm layer of filter gravel. Should I use "free drainage" or "seepage face" as my lower water flow boundary condition?

Bo

Hi,
This depends on wat bottom boundary condition you want.
I would take the free drainage because this fits the actual situation the best?, but if you need horizontal streamlines I would choose deep drainage. But it depends on wat bottom boudary you want to have.

IJsbrand

## ID = 150, Long running time

Dear all
I'm facing a problem that any of you may have encountered, and may be able to give me an advise:
I'm simulating water infiltration through a layered soil, exposed to atmospheric conditions and initially almost saturated. When running the software, results won't converge - i.e. the number of iterations performed is above max.
Results converge only when I decrease the initial time step to 0.000001day and I allow a max iteration number of 100. The simulation is to be run over 365 days. So as you can predict, the running time required is practically infinite: I'm not able to run it because it would take forever!
Any advise?

Sophia Ghanimeh

Sophia,

I will recommend here several variables that I believe are optimal for seasonal (long-term) simulations.

Theta tolerance: 0.001
Head tolerance: 1 cm
Initial time step: 0.01 d
Minimum allowed time step: 1e-6 d
Maximum allowed time step: 1 d
Discretization close to the surface: 1 cm
Discretization at larger depths: 5 cm (not on boundaries between layers).
If you have clay soils, use -2 cm air entry option.
hCritA= 150-500 m
If you have course soils, decrease hCritA to less than 100 m.

With these or similar parameter combinations we have successfully run hundreds of simulations for simulation times of hundreds of years.

Jirka

Hi Jirka and all,
Almost a half year ago I asked you the same question regarding a simulation I wanted to do for 10 years. I also came to a simulation time that was langer than 10 years. (field observation is in this case better... ha ha). I used the parameters Jirka suggested (then and now again). But I have also a little better results in reducing calculation time by very localized refinement of the mesh and understanding of were the (computional demanding) steep gradients were present. I had very much problems with a thin layer of (20 cm) clay. Furthermore, I also had an inpermeable layer present but this was not the real problem.
The end result was that I had to tell my financer and commisioner that this simulation was not practicly possible. He agreed fortunatelly.

O also a stupid trick that works but it is not correct:
If you have a rainfall event that starts/ends abruptly this is also computional demanding. If it is not a problem you can change this by letting it increase/decrease slowly.. You can keep the total amount of water be the same. see example below.

0 change to 0
0 ,, ,, ,, 2
0 ,, ,, ,, 10
52 ,, ,, ,, 25
0 ,, ,, ,, 10
0 ,, ,, ,, 5
0 ,, ,, ,, 0

IJsbrand de Haan

Dear IJsbrand

First, concerning the finer meshing of high gradient areas. Do you think that a fine mesh is necessary at soil surface under atmospheric conditions?
Second, about replacing the abrupt change in precipitation with a slowly increasing pattern, I find it a good idea: I'll try it. Yet I don't know why did you say that it's wrong???

Sophia


Dear Sophia and All,
I am also running long-term (12 yrs) infiltration simulations through a strongly layered soil and having similar problems that Sophia is facing. Following Jirka's suggestion, I used the parameters below, but that did not help very much. I decided to construct a 1-d problem using Hydrus-1d and used the same atmospheric boundary conditions I used in 2-d problem. After setting a very fine discretization near the surface, I observed that convergence problems occur during heavy rainfall events. I followed IJsbrand's suggestion about setting slowly changing rainfall values (total amount of water the same). I was able to eliminate a number of convergence problems by doing so, but for one event this did not work. The code gave the following error: floating point error- zero divide!

For this particular event, I actually got rid of some of heavy daily rainfalls to understand the origin of the error; the program did not stop, but generated the same error during the subsequent heavy rainfall event.

I am wondering if any of you had the same error while dealing with convergence problems. In this case, can say that this error is mainly due to convergence problems?

My current solution is to adjust rainfall amounts by setting slowly changing values whenever the program runs too slowly or stops because of the error. Since I am running for a long period of time, this method is very inefficent since I have to change the values manually. Besides, I would rather keep the original values if I know how to deal with this problem differently. Any suggestion will be appreciated.

Sevim


hi all,
I want to comment to "my rainfall trick" I think it is wrong as it is not actualy happening in the atmosphere. If you wan't to create a valid simulation you have to stick as close to the actual input data as possible. And about the fine mesh at the surface I didn't try it yet maybe it helps. Let's try it.

But maybe we can ask Jirka why hydrus has so much problems with steep (abrupt) changes in the rainfall input? Or maybe it is only a problem when used in combination with less permeable soils?

Furthermore some thoughts about the unsaturated hydrology : I think there is a sort of smoothing (relaxing) function in the topsoil layer because the abrupt rainfall events

are transported more smooth to the underlaying soillayers. Also the ponding mechaninism is not included as a "smoother" is it? What is the forum thoughts abot this?

I am wondering if this is a well known problem and already described somewhere? (?)
greetz,

Hi,

As far as I know HYDRUS can have problems to converge for high intensity rainfalss for soils with very low n values (<1.1). The reason is that the Mualem model predicts unrealisticly fast decrease of the hydraulic conductivity close to saturation (and correspondingly large nonlinearity). Somewhere else on these pages I have discussed that and gave a reference to manuscript of Vogel et al. (2001) that deals with this issue. One can simply overcome these problems by selecting van Genuchten model with air entry value of 2 cm (which decreases nonlinearity of the problem). Then the simulations are smooth and fast.

Jirka

Vogel, T., M. Th. van Genuchten, and M. Cislerova, Effect of the shape of the soil hydraulic functions near saturation on variably-saturated flow predictions, Advances in Water Resources, 24, 133-144, 2001.

Hello:

I am a new user at PC progress forum.I am trying to use hydrus 1D to simulate infiltration through a 100 cm clayey soil column overlying a water table. The initial pressure head conditions are a negative pressure between 228cm at the top surface to 0 at the bottom of the column. The boundary conditions are steady in filtration at the top face, and zero head at the bottom face.The simulation time period is 100 hours,with inital time step at 1 hour and min time increment as 0.0001 hours and maximum time increment as 100 hours. I am using all other default parameters.The solution does not seem to converge even after entering the air-entry value option. Do you have any suggestions?
Jaspreet

Jaspreet,

Keep in mind that fluxes in Hydrus-1D are negative downward. Do you specify negative prescribed flux? Check that.

If that does not help, zip the folder with input files and the *.h1d file and send it to me. I will have a quick look at it. This should work without any problems.

Jirka

Jirka,

The solution now converges as I put a negative value for flux.
Jaspreet

Dear Sophia,
beside the given remarks, my experiance (simulating top soil with ks< 10-7 ms and high precipitation events) was that

hTabN = 0
hTab1 = 0

gave more stability.
Have also a look at the chapter 6: trouble shooting in the RASSAM etal Hydrus book, there are some good hints.

regards – Uwe


## ID = 151, Error message

Dear all
I'm receiving the following error message during execution (i.e. during the running time): "exception floating-point overflow"
Does anybody know what it means or what might be the cause?
Sophia Ghanimeh


Sofia,
mar7aba
I believe that the "floating point overflow" error message is the only message that appears when your program can't run. You will have to go back and check all your parameters and your Iteration Criteria. Make sure that your discretization is adequate according to the type of soils you are using.

You can wait for Jirka to clarify this to you.
Mazen


Hi.
I try to simulate the migration of water through the multilayered protective barrier of a waste disposal.
When I launch Hydrus-2D, it begins normally but an error message appears:

Sensitivity of protective barrier performance
Vertical plane flow, V = L*L
Reading nodal information
Reading element information
Reading geometric information
Reading material information
Generating materials
Reading time information

Floating Point Error – Invalid number !

Error when writing to an output file !

Press Enter to continue

Does someone know someting about this message?

Hi Benoit,
Make sure that your time interval is adequate (usually very small) fro you discretization. I usually use default setting of HYDRUS, the only thing that you can manipulate is initial time step and time interval.


Dear all (maybe specific Jirka),
Does anyone know if it is possible to copy a list of all simulations and the description of the simulations.
It is nicely listed in the project manager [thanks codecrackers]. But I wanted to tell the world what simulations I did and I retyped a part of them manually (a lot of work). I used also a screen dump but this does not look very professionally. The list of the names of the simulations and there description is stored in some file but which and can I read as a text file (then my problems are solved.)
When I can get the list I can change my screen dumps (ya, more than 6) because the windowsize can be altered.

IJsbrand




Hi Benoit,
Make sure that your time interval is adequate (usually very small) fro you discretization. I usually use default setting of HYDRUS, the only thing that you can manipulate is initial time step and time interval.



Hi mizo316,
I thank you for your advice but it seems that I have just found what was wrong. I think that the problem came from the building of the domain: I inserted independant points on a line of the domain. I had no error with Meshgen (like it occurs when you cross lines).
When I removed these famous points, this error message does not appear anymore (another has appeared but I would like to think before speaking about). However, it's curious because I believed that it was permitted. Or it's a coincidence...?!
Benoit.


Hi Benoit,

When I started using HYDRUS, I wasn't careful about the individual points. It is kinda tricky especially when the geometry is ok! You only learn to run simulations the hard way (by keep trying). Each time change ONLY ONE parameter and monitor the difference in output.

Mazen


## ID = 152, RETC vs. Rosetta

Hello everybody,
I would like to ask you what your experiences are with Rosetta and with RETC respectively. I have done a parameter estimation with both programs and the results are quite different. It's hard to see for me which results are more reasonable as the input data is completely different for both programs.

In Rosetta I enter the textural classes, BD, and WC at 33kPa and 1500kPa. The calculated Theta s and Theta r are both underestimated quite a bit. Alpha is about 10 times smaller than the alpha calculated by RETC (where Input is Theta s, theta r, and 8 retention curve data pairs), and the values of n are estimated to be about 0.3 bigger than calculated by RETC.

I think the main problem is that my data are somewhat untypical: The measured values for 5 different soils (loamy sands, loams, and sandy loams) have measured values WC of 0.43-0.68 at 1kPa and 0.12-0.20 at 1500kPa. (- the percentage of organic substance it up to 17%).
The reference values by Carsel and Parrish for these kinds of siols would be Theta s = 0.41-0.43 and Theta r = 0.065-0.095.




So here are my questions at last:
Does anybody have any experiences about which program, RETC or Rosetta, gives better results for alpha and n?
Does anybody know which program would give better results for soils with a high percentage of organic substances?

On the one hand I am tending towards believing RETC because it seems more reasonable to me to use all the available retention curve data as input. On the other hand I don't really know the exact Theta s and Theta r values and RETC is quite sensitive to changes in the Theta s input. It's sort of a dilemma...
Klaus


Hi klaus,
your question is quitte interesting. I have worked with both program's. I worked already with RETC before it was a dos-program so I know this program best. The old program was written to work on a sun workstation with therminals, it was more versatile and you could use every data you had as input. In the newer versions this was reduced to a minimum data set that was required to obtain a reasonable curve. In

general how more data you put in your pF curve fitting program the more acurate curve you get. Rossetta is based on some more general soil caracteristics. It is based on a large database with many soils which is used for fitting. (if i am not mistaken). When I had a lot of data and thought it was more importand that the curve was fiting the data than I fitted the data in a Excell sheet. Plot the data point and the curve and change the MvG parameters. If you think you have enough data to obtain a pF curve manually do so. I had to do manual fitting as I wanted to create pF curves of non-soils!

Furthermore both programs are based on the assumption that there is no or only a little organic matter. i.e. The pF curves are unique for one soil. The programs use the grain size distrubution as main data input and this not the waterholding capacity for organic soils.

Which is the best output I can not say. compaire the measured data points with the curves the best fitting is probaly the best curve.If you want a generalized dataset compaire with other data sets of soils alike.
IJsbrand de Haan
Hi IJsbrand,
thanks for your advice. How many retention data pairs do you think are necessary to fit a curve manually?
Klaus


Hi klaus,
The amount of data points is less importand than the range of were you have your data points. I think you need about ten points evenly diveded over the curve. But the problem is; you also need points near wilting point and near the first en the second curve. You also need to know the Theta S well.
If possible you should measure several samples to obtain an averge of 1 soil. Comparison between curves is also possible and a you get a general idea of the spreading of your data or the uniformity of your soil.
Do you know the multi-step method to obtain pF curves? (if not you can find it on the internet).

IJsbrand de Haan


Hi IJsbrand,
thanks again. It is quite a clear story, i think my data might be sufficient, although Theta s is not as clear as I'd like it to be. I'll still try it though, at least to have a comparison.
best regards
Klaus


### ID = 153, Problems shifting to Win2000

Hi out there!

We are shifting to Win2000 and are experiencing kind of severe troubles:
I downloaded the new Installation program from
http://www.ussl.ars.usda.gov/MODELS/MODELS.HTM
as indicated in the FAQ. After installation I overwrote the pcpinfor.sys and everything seemed to work well, at least with the examples. When I tried to open my own workspace created under WinNT/Hydrus 2.008, but the programme would not read or recognise the 2d-files. When trying to start hydrus by opening h2d-files directly, sometimes obscure errors regarding meshgen were reported, however I could not find any systematic behaviour.

If anybody experienced any similar problems when shifting to Win2000, I would appreciate any hints.

Hector

Hi Hector,
the installation on http://www.ussl.ars.usda.gov/MODELS/MODELS.HTM is probably old (has not been updated). Please try to reinstall Hydrus2D and use installation 2.008 that you can download from our WEB site:
http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm

I'll have to fix the text of FAQ #16.
Regards Mirek

Hi Mirek, ty vole!
Thank you for your rapid response. I downloaded from the site you indicated and installed again (... our system operator will think I am trading mp3 from the volume I am handling at the moment ;-). Now at least a new workspace can be created, however Hydrus does not recognise the existing projects therein with exception of a copy of one of the example files! Is there something new, the way Hydrus resolves the information in the h2d-files? Do I have to assign any rights to those files? Pleaaaase, we have spent too much time on installation.
Hector

Hi Hector,
all your project files should NOT have the "read-only" attribute and all project directories should NOT have "read-only" or "archive" attributes. Please check it.
Mirek

Hector,
With Rien we often travel to give HYDRUS short courses. We never encounter any problems with installations, when installed on individual PCs (with any type of Windows 32 bit operating system, i.e. Wndows 95 or later, including NT and XP) and when the user itself has all the authority over the PC. We almost always encounter problems, when IT departments get involved. They often remove from user the authority to handle their own PCs and that causes problems.

For example the ini files of HYDRUS are stored in the windows\system folder and it must be possible to modify them, to update information about existing workspaces and projects. Thus there is a problem if this is prevented by IT people, which is almost always the case in computer rooms for students.

Also, all the input files and *.h2D files cannot be read only (which is often after copying from a CD). HYDRUS project thus need to be stored somewhere where the user has a complete authority over what he does (without involvement of the IT department policing - Big brother did not go away in 1989!).

Jirka

Hi Mirek & Jirka,
thank you for the hints. After removing the archive-bit from the project-directories and h2d-files the programme was able to resolve the projects in the workspace menu. Interesting however, one could open projects via the open file option before removing the archive bit.... So anything seems to function on Win2000,

Thanks again.
Hector

Hi Hector:
The problem caused by "archive" attribute of directories is actually a bug in Hydrus project manager. There is no reason why project directories should not be allowed to have this attribute. I've known about this problem and I'm going to fix it in next installation (2.009). Fortunately this error appears very rarely because directories usually don't have this attribute. Project files can have the "A" attribute because the error concerns only directories.

Regards
Mirek

**ID = 154, Mesh generation errors**

Dear All,
I want to see the effect of deep drainage on groundwater table, so I have a domain with 400 m long, 8 m depth (upstream), 4 m depth at downstream with deep drainage. I defined the Mesh by drawing a line and point manually.
The problem occurs when I click "make mesh", it said that "Hydrus2D.exe has generated errors and will be closed by windows"
Zal

Hi Zall,
I had a look at your project and on my computer I didn't get any break-down or error. However, the generated mesh was very irregular and inappropriate for Hydrus

calculations. Your computational domain is long and narrow but it should not be any problem for Meshgen. I think that the main problem is in the way that you are trying to stretch the mesh in horizontal direction. You have defined many horizontal internal lines in the computational domain - why? Also there are almost no mesh points on the outer boundary curve and therefore Meshgen can not create a regular triangulation that would respect given internal curves and just a couple of points on boundaries.

I'd recommend you to:
1/ Delete all internal curves
2/ Edit mesh densities at boundary points and use
different values for "left" and "right" directions
3/ Edit mesh parameters and define appropriate
"Mesh stretching" factors

Have a look at an example of FE-mesh generated in your domain
http://www.pc-progress.cz/_documents/Hydrus_Meshgen_01.pdf
The mesh is stretched horizontally with factor 1:30 but you can define your own factor. Of course you can use internal lines if you want but I'd recommend you to start without internal lines as the first step.

Regards
Mirek


Thank You Mirek and All
Now I am able to finish my Mesh, But unfortunately when I am running the calculation, there is another error appears. It said that " ORTHOMIN termites-too many iterations, time step reduced."
Zal
Zal,
Mirek sent me your project (including mesh, initial and boundary conditions). The mesh is completely inadequate for your problem and its size. Many parameters do not make sense. You need to first study problems of numerical modeling of the vadoze zone, what kind of meshes to use, what values of different parameters should be, and so on. That's why we provide all those examples that come with the HYDRUS CD. Look, for example, at the example Plume, which is for a larger scale. Or get a HYDRUS book, which guides users with most of the steps needed to successfully run HYDRUS.

Jirka


Dear all,
At the present time, I work on the article of Dirk Mallants (et al.) about the capillary barriers (Mallants, D., G. Volckaert, and J. Marivoet, Sensitivity of protective barrier performance to changes in rainfall rate, Waste Management 19, 467-475, 1999.).
In a first step, I wish to get back their results and conclusions but it does not work properly.

I have recently launched a simulation but regularly, I see on the screen: "ORTHOMIN terminates - too many iterations, Time step reduced". I do not interrupt the calculations, not knowing if it is really problematic.
So, I have some questions:

Is it really an error message or is it simply an informative message to say that the computer re-calculates with a smaller time step?

What is the link with the number of iterations displayed on the screen (because on the screen, it takes 2 or 3 iterations. I do not see why the number of iterations would increase so abruptly that ORTHOMIN terminates)

If someone encountered such a problem and solved it (Zal?), can he say me how?
Benoit.
Benoit,
The number of iterations printed to the output run_inf.out and reported on the screen is about the number of iterations during one time step, i.e., in the Picard linearization of the Richard equation.

Orthomin is an iterative matrix solver (system of linear equations). When the matrix is not well conditioned, the solver can fail to find the solution. The smaller time step is then used to assemble the matrix again (that is the message you got, and if teh code works fine with smaller time scale, your calculations should be OK). This typically happens when there are very contrasting regions in the system - perched water layers and dry regions.

Your questions:
a) Correct. The code simply tries with smaller time step.
b) above.

Jirka
Dear Benoit
I faced this problem many times, espescially when I had large differences in the hydraulic conductivity of the cover layers. I used to solve it by refining the mesh (reducing the element size) at interfaces or using smaller time steps.

You can try it.
I tried a capillary cover myself so if you like to discuss it you can e-mail me on: sghanimeh@ndu.edu.lb
Sofia


Hi,
Two error types seem to occur in my simulations:
The first one consist of a "floating point error". It appears when I ask a fine tolerance (h tolerance < or = 0.1 cm). I wish to solve this problem rigorously (without changing one parameter after another and see if it works). To achieve it, I tried to calculate a stability criteria between the spatial and the temporal discretization but without any success (because of the non-linearity between "theta" and "h"). Have you already calculated a such criteria? If it is indicated in the manual, sorry: I did not see it.

The second one is not really an error but seems curious to me: when the simulations terminate properly and that I have a look at the free boundary flux depending on time, some oscillations appear after 50 days. The error stays under 1% but I have no much confidence in it. What do you think about?

I know that other topics have already treated the problems but the answers do not satisfy me.

I have a comment about the way the time is printed on screen and in the files. When the time step is less than 1e-04, all the calculated values correspond to the same time value (the times 0.00012, 0.00015 and 0.00018 are refered to 0.0001 day): it is not really problematic but it looks like a stair when zoomed in. I suppose that the values considered in the calculations are more accurate.

Benoit.


**ID = 155, Feddes plant model**

Are values known for the Feddes plant parameters for
trees (I can email specifics if needed) used within hydrus-2d ?
I noticed many of the parameters have the same value
in the database across the board, so I would like to know how
these parameters were derived in the database
(i.e, have they realistically reproduced plant uptake and or
lateral movement of water in field settings within/without
hydrus-2d in your application(s) ?


The database used in HYDRUS codes for parameters of the root water uptake stress function is taken from:
van Dam, J. C., J. Huygen, J. G. Wesseling, R. A. Feddes, P. Kabat, P. E. V. van Walsum, P. Groenendijk, and C. A. van Diepen. 1997. Theory of SWAP version 2.0, Report 71, Dept. of Water Resour., Wageningen Agricultural University, Wageningen, the Netherlands.

and there referenced Wesseling [1991] and Taylor and Ashcroft [1972].

I'm not aware if these parameters are available also for trees.
Jirka


dear rajan,
You need to get better data than provided by the sources you described. Very offten they only are valid for non-deep rooting crops (approx 1 meter minus surface). I have some literature on forrest hydrology but I dont know if they contain actual Feddes parameters.
If you know enough of the trees (wilting point etc.)and their rooting depth you can determine Feddes parameters yourself. I hope the literature helps:

Bosveld, F.C., 1999. Exchange processes between a coniferous forest and the atmosphere. Wageningen University. Ph.D thesis.

Bouten, W., 1992. Monitoring and modelling forest hydrological processes in support of acidification research. University of Amsterdam. Ph.D thesis.

Dolman, A.J. & E.J. Moors, 1994. Hydrologie en waterhuishouding van bosgebieden in Nederland, Fase 1: Toetsing instrumentarium. Wageningen, Staring Centrum, Rapport 333. (hydrolgy and waterssystem of forrest-area's in the Netherlands, Phase 1: Testing instruments[ydH])

You can have a look at the alterra site (www.alterra.nl)to exploire the knowledge available yourself.

IJsbrand de Haan

IJSbrand,
The website link was in dutch. Would it have the phd materials
there ? Tree are quite complex relative to crops so standard
notions of "wilting point" etc. may not be readily applicable...
Ranjan

dear rajan,
You need to get better data than provided by the sources you described. Very often they only are valid for non-deep rooting crops (approx 1 meter minus surface). I have some literature on forest hydrology but I dont know if they contain actual Feddes parameters.
If you know enough of the trees (wilting point etc.)and their rooting depth you can determine Feddes parameters yourself. I hope the literature helps:
Bosveld, F.C., 1999. Exchange processes between a coniferous forest and the atmosphere. Wageningen University. Ph.D thesis.

Bouten, W., 1992. Monitoring and modelling forest hydrological processes in support of acidification research. University of Amsterdam. Ph.D thesis.

Dolman, A.J. & E.J. Moors, 1994. Hydrologie en waterhuishouding van bosgebieden in Nederland, Fase 1: Toetsing instrumentarium. Wageningen, Staring Centrum, Rapport 333. (hydrolgy and waterssystem of forrest-area's in the Netherlands, Phase 1: Testing instruments[ydH])

You can have a look at the alterra site (www.alterra.nl)to exploire the knowledge available yourself.

IJsbrand de Haan

Hi,
There is also a english link there but maybe this is interesting:

http://www.alterra.nl/english/publications/publicatielijst2.asp
I know that there has been research on forests, SWAP, modeling and hydrological characterisation on the Alterra Institute. You have to explore yourself. try otherwise the internet

greets IJsbrand

## ID = 156, Help for a beginner

Hi
iam a recent hydrus user and am having difficulties in getting going and wondered if you could offer some help
My intention is to simulate water flow in soils using data i have already acquired to see if the simulated matches my experimental. My columns are cuboid is shape with dimensions of 20 x 20 x 20 cm. The columns were subject to an infiltration at a rate of 10 mm hr until saturated. This experiment lasted 24 hrs during which 3 tdr probes inserted at the top middle and bottom measured vol water content at 1 min intervals. We also have texture, bulk density, chloride breakthrough data for the soils. At a point in the experiment we infiltrated with brilliant blue dye and after the experiment scrapped away soil layers to reveal the dye pattern so we could examine the water flow mechanisms. We want to compare this with simulated flow patterns derived from hydrus for our different soil types.

I have gone through the tutorials and help options but iam really struggling to perform this task. In particular,
1) Iam unsure how the rectangular geometry and space discretization section work - the help function is apparently unavailable for these. I assume they reflect the size of my sample but iam unsure of the inputs.

2) I assume at some point I need to input my TDR data but cant see where this goes

3) I cant find any reference to the water input for a given flow experiment but surely this is key in terms of rainfall intensity and duration - am i missing something ?

I appreciate that these comments are probably straightforward but believe that i have spent the last week going round in circles so any help would be much appreciated.

SJM

Dear Sacha,
It seems that your experiment is just ideal for simulating with HYDRUS codes. It depends how you want to proceed. There are two HYDRUS codes available at this moment: HYDRUS-1D for one-dimensional problems, and HYDRUS-2D for two dimensional problems. Thus in neither of them you can do a three dimensional flow and transport. You need to conceptually simplify your problem into one- or two-dimensional problem. Most column studies are usually analyzed as one-dimensional problems.

Look at some examples distributed with the code. They can certainly get you started. Test1(workspace Direct) - simulates one-dimensional infiltration from ponding. Thus, that example seems to be close to your problem except that you have flux upper BC, which is easy to change.

Measured TDR data: Look at example HorInf (Inverse) which is an inverse problem with measured water content data.
Test2(direct) - example of simulation with different rain intensities.

Jirka


## ID = 157, Performance enhancement with air-entry option

During a series of runs for a systematic analysis of a simple seepage Problem formulated by Polubarinova- Kochina (immediate drawdown at the boundary) I encountered a much faster behaviour applying the 0.02 m air entry value option. Some computations would not converge at all without the air entry application.

I just wonder what is the reason for such a performance enhancement, the retention curve and the rel. cond. Function remain basically unmodified, only the capacity function is forced to zero when reaching the a.e. point. Anybody has any explanation for what really happens at low suctions? Why does this minor mofification in the parametrization has this effect? Has something to be considered, when applying this option?

Hector


hi hector,
I just want to add a question:
Does the simulation contain a clay soil?
In this soil type it really makes a difference.

IJsbrand


Hector,
Effects of the air-entry value of 2 cm:
a) The effect at the retention curve is almost negligible, especially for fine textured soils. You cannot practically see any differences if you plot the two functions.
b) The effect is significantly larger on the unsaturated hydraulic conductivity function. The function decreases significantly slower with decreasing water content (pressure) than the original function (see Vogel et al., 2000). This is the reason for much faster drawdown.

Jirka

Vogel, T., M. Th. van Genuchten, and M. Cislerova, Efect of the shape of the soil hydraulic functions near saturation on variably-saturated flow predictions, Advances in Water Resources, 24, 133-144, 2001.

Hi
I just want to share what I concluded from previous readings. Hope it's right.
When the air-entry value is set at -2cm (i.e. the medium is considered constantly saturated at suctions below 2cm) the problem becomes more numerically stable near saturation. Especially for clayey soils having a highly non-linear conductivity function near saturation.
Sophia

Correct.
J.

## ID = 158, HYDRUS with Win XP ??

Does any body experience problems running HYDRUS with Windows XP?? as the newest version (2.008) is for Win95,98,NT.
I can not run the calculation after upgraded my PC to Win XP.
Thank's
Zal

Hi Zal,
Hydrus 2.008 runs under WinXP. I don't know exactly what problems you have - what message do you get when running calculation?. Also have a look at the topic "PROBLEMS shifting to Win2000" (http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=153).

Regards
Mirek

Dear Mirek,
After I checked and re-checked again, I found the answer of my problem on FAQ 26. After I save my project file on "HYDRUS" folder, I can run the HYDRUS calculation now.
Sory about my stupidity.
Regards,
Zal

Hello all,
I have ran several long problems, and a full project, on Windows XP. It appears to run just fine.

XP did cause some confusion for me at first where Hydrus would not run. Moving the files to the Hydrus directory also solved my problem.

However, I do not think the files needed to be moved to the Hydrus directory. I think the problem is that the file pathway in most XP directories (especially my documents area) is too long. I am guessing that there is a character limitation in the Hydrus software that is exceeded if a very long pathway for the input files is used. The software would have to write an even longer pathway to link the locations of the exe file with the input file.

Hi,

The graphical interface can handle paths of any length. There is not limit on the number of characters (at least I think).

However, the computational module has a limit on the number of characters in the path, and it is 100 (see below).

character cFileName*100,cDataPath*100

Newer versions of our software will give a warning when you start running the computational module.

Jirka

## ID = 159, Boundary flux

Dear all
I'm modeling a domain having three seepage faces: two sides + bottom.
The results in Cum_Q.out represent the cumulative flux of the three seepage faces combined. Yet, I need to determine the cumulative flux along only one of these faces (the bottom one).
Do you have a suggestion on how to find it or calculate it? In other words, how to separate the flux values of every seepage face independently?

Thanks
Sophia Ghanimeh

Hi Sofia,
One way to do that is to add the flux acroos all the nodes of the surface you wish to find the total flux across it.
Mazen

dear Sofia,
I don't know if it works but maybe you can add an observation point near or on the boundary your interested in? I don't know for sure if you can see the accumulated flux of an observation point? (never needed it)But you can give it a try maybe..
greets IJsbrand

Sophia,
In the output file Boundary.out we print at selected print times information about all boundary nodes, such as pressure heads, water contents, boundary fluxes, and so on. To get a flux for any part of the boundary, you can simply add fluxes in corresponding nodes.

See also description of this problem in David Rassam's Hydrus book.
Jirka


**ID = 160, Hydrus1D, temperature dependence, mass balance err**

Dear Hydrus Users and Developers,
I am running Hydrus-1D, Version 2.01 to simulate pesticide fate in soil, based on field and lysimeter experiments.

Recently I introduced temperature dependence of Reaction Parameters (SinkWater1, SinkSolid1, SinkGas1) and Henry Constant. Since then my mass balance errors increased up to 100%.

From the earlier postings I learned, that there have been some problems, but I could not solve my problem with this information.

Is there some similar experience and maybe some hints or solutions to this problem?
Roman


**ID = 161, Velocity vectors**

The color scheme generated for the velocity vectors doesn't
always come out that well in B&W reproduction. What
approaches would you recommend for getting B&W images
of the velocity vectors ? It would be nice if there's
someway to put the absolute values on the tip of the
arrows (there are probably better ways, ex. through matlab).


Ranjan,
you can use another color palette or B&W palette: go to the Graphics->Options->Color Palette dialog and select "Standard, 1c, black" as the current palette. All lines/vectors should be black => they will not be visible on screen with black background but if you use "Print" or "Copy to Clipboard" functions you will get B&W printout. You can also create your own palettes.

Unfortunately displaying values is not possible in the current version. We register all good ideas that we'd like to add in a next version.

Regards Mirek

**ID = 162, UK Hydrus users ?**

I am a PhD student in Nottingham struggling to get to grips with Hydrus2D. Just wondered if there are any UK Hydrus users who might be prepared to spend a few hrs (if I visited) to help get me up and running asap. Just need a few pointers in the right direction with regard to simulating water flow in a 2D saturated soil column. Any help appreciated !

I suggest that you contact Steve Buss at

Environmental Simulations International
Groundwater Consulting and Software
Priory House, Priory Road,
Shrewsbury SY1 1RU UK
Telephone +44 (0)1743 280020 Fax +44 (0)1743 248600
email <mailto:SteveBuss@esinternational.com>
web <http://www.groundwatermodels.com>

Last year (2002) they organized a Hydrus short course for us and thus not only have experience with the code, but also can provide you with a list of people who took the course.

Jirka

**ID = 163, Flux at nodes**

Dear all
I need to find the cumulative flux as function of time across a number of nodes inside the domain. I tried to set them as observation points. It seems that I can only get the head and water content at those points - I'm not considering temperature or solute transport.

Do you have any suggestion on how to come up with the flux-versus-time graph at an internal node?

Sofia

Hi Sofia,
If the nodes belong to a defined boundary (constant head, seepage face, etc..). You can go to the boundary.out file and look up the flux values versus time. I am not sure how to do that for any random node in your matrix.
salam,

Dear Mazen

Thanks for replying. The problem is that I need to find the flux across an internal interface (a string of internal nodes).

Any suggestion??

Sofia

Sophia,

Hydrus does not have an option to calculate fluxes across internal lines. However, I have recently developed a routine that does just that. But it requires additional input (nodes and elements on the line) and that is not supported by the graphical interface and needs to be prepared manually. If you are interested in such update, send an email directly to me and will send it to you.

Jirka

## ID = 164, Which reference temperature in Hydrus-1D?

Which is the reference temperature in HYDRUS-1D?

Is it 20°C (273.15 K)?

Roman

Sorry, I mean:

Is it 20°C (293.15 K)

Roman

Which is the reference temperature in HYDRUS-1D?

Is it 20°C (273.15 K)?

Roman

Correct. It is 20°C, i.e., 293.15 K.

Jirka

## ID = 165, Boundary & initial conditions for upper nodes

Hello,

I am trying to use HYDRUS-2D in an inverse modelling framework in order to determine the soil hydraulic properties from a single ring infiltrometer test.

My problem has to do with the inital and boundary conditions window. The infiltrometer is represented by a constant head boundary at the upper surface of the soil/mesh. Obviously the corresponding nodes have both head and initial water content values which I cannot translate in Hydrus i.e if I start by specifying the initial water content, the head is automatically set to the same value. And if I start by specifying the constant head value, the inital water content is changed. It's a bit of a catch 22.

A friend suggested using a variable boundary condition starting the 2nd time step. Apparantly this worked for her simulations in direct mode. However, I am having problems dealing with it in the inverse mode.

Has anyone encountered a similar problem and more importantly found a solution? :)
N.


Nouchka,
Suggestion of your friend was a good one. That's the way it needs to be done. You can look at the example in the "inverse" workspace. It is for multiple pressures, but it workes similarly for one pressure as well.
Jirka


hello nouchka,
Do you know that the single ring infiltrometer has some severe limitations? The flow is horizontal and vertical. The area where the flow is occuring is not known and therefore also the flowregion. There is a double ring infiltrometer invented to limit the horizontal flow.
I know I made the problem worse, real devilisch ... but better save than sorow
Or are you trying to use Hydrus to arive at better preditions of the SRI for the permeability ? In that case the above is not appllicable..


Hello IJsbrand,
Actually, I am trying to use Hydrus to estimate conductivity values which I will compare to those obtained by double ring infiltrometers and rainfall simulators at the same location. I have also Ks values calibrated using 5 rainfall/runoff models. I think that all these provide good "prevision bounds" to compare my results with. Do you not agree?
N.

Hello Jirka,
What type of data are you specifically looking for? Do you need simultaneous measurements? What are the spatial scales and precision that you have in mind?
Let me know what you need.
Good day,
N.


Rainfall intensities, surface outflow, soil hydraulic properties, instrumented soil profile (tensiometers, TDR?), surface geometry, preferably homogeneous (not needed). I do testing runs on 50 m transect, but obviously experiment may be smaller.

Jirka


### ID = 166, Effect of hCritA on underlying soil

Dear all
I'm modeling a two-layers soil. The top layer consists of vegetative soil and the bottom layer is clay, with big difference in hydraulic conductivity and n value.

From the soil-water characteristic curves (SWCC), at h=-100m the vegetative soil dries with a water content equal to residual (which is why I have chosen hCritA=100), but at this suction the clay still have water content of about 0.27 (Which is close to saturation).

From real life experience, both layers should dry during the summer season.
However, the evaporation process stops when the surface head reaches hCritA (-100m). How can the clay dry if the evaporation stops???

I tried to increase hCritA to 500m, I got a runtime error: the program became numerically unstable.

I tried to change the n factor in the SWCC of clay, the drying process partially started. But this is not a solution because I'm changing the nature of soil (the clay SWCC becomes very similar to that of the topsoil)

Sophia Ghanimeh


Sophia,
If the surface layer reaches the residual saturation and thus negligible hydraulic conductivity, the clay below the surface layer will never get dry. That's the quality of the layered profile that is being very often used in engineering. One spreads a course sand or gravel at the soil surface and that will prevent evaporation. Since the conductivity of the surface layer is very small, evaporation can proceed only by gas diffusion across the surface layer (which is small and often neglected. And not considered in Hydrus).

Jirka
One more thing, do you have a suggestion on what exactly can be modified in the top layer, in order to initiate the drying of the underlying clay? For instance, using a higher hydraulic conductivity, higher n value, etc?
Please note also that the top layer is only 15cm deep.

Regards
Sophia Ghanimeh


Sophia,
If you want to initiate drying of the underlaying clay, you need to use fine textured material at the surface. Once you have course textured sand or gravel at the surface, underlying clay should not get dry due to evaporation.

Jirka



**ID = 167, Huge output files**

Hello everybody!

I'm encountering a strange thing right now: I switched back to Windows 2000 because windows XP didn't run properly on my PC. Of course I had to reinstall HYDRUS 2D, but that wasn't a problem.

But: suddenly the output files of the new simulations are huge! teh Cum_Q.out ifle has around 90MB and the ObsNode.out file has got even 110MB for only 5 ObsNodes. v_mean, h_mean and run_inf are also very big.

does anybody have any idea what can be the reason for this? I don't think it's the simulations, they are very similar to the ones I ran last week on Windowas XP.

thanks for any answer,

Klaus

Klaus,

I do not see any reason why the output files should be different sizes on PCs with different operating systems. The files are formatted and thus should have the same size. The only reason can be that you altered the problem and now you have a very large (incredibly large) number of time steps. Open those files and see what's in them.

Jirka

Hello Jirka,

thanks for your reply. Yes, you're right, there are incredibly many time steps, and yes, you're again right, I altered the problems I am simulating, but only slightly. For the most simulations I changed the initial water content. But I wouldn't have posted this "problem" if these huge output files had happened only once in a while. Thing is that since I have reinstalled Hydrus, ALL simulations create such big output files. And I don't have any idea why the programs needs more timesteps for the calculations now. I did not alter the time information and the iteration criteria.

Do you think it might help to again reinstall the program?

Klaus

Klaus,

here is no reason whatsoever, why the code would take a different number of time steps under different operation systems. The computational code is written in Fortran, which as standard language, fully transferable from one OS to another. If you do not want these huge output files, when you need to use a lot of time steps, you should print the output only at specified print time (deselect "T_level information" in "Print information"). The output is still printed whenever the BC is changed (new BC record read) into the file A_level.out. Look at this file using spreadsheet (Excel).

I'm working on a version that will print after specified number of print times or at requested intervals. It will be available soon.

Jirka

**ID = 168, Non-vertical axis of symmetry?**

Hi everybody
I'm wondering about the possibility of modeling an inclined surface using axisymetric option. In other words, is it possible to define a non-vertical axis of symmetry?
Sophia Ghanimeh

Sophia,
No, it is not. The axis of symmetry must coincide with the direction of gravity. If it does not, then the problem is not axisymmetrical.
Jirka

**ID = 169, Inverse solution of ring infiltrometer test**

Hi!
I am trying to calculate several inverse solutions of soil hydraulic parameters from ring infiltrometer test data. The model is very simple:
1 homogeneous material, vertical flow, constant pressure (=0) top boundary and variable pressure bottom boundary, no flux vertical boundaries. The data for inverse solution consists of time/flux measurements.
Some calculations go well, but some just crash for no apparent reason. It has something to do with the soil hydraulic parameters, but I can't figure out what. Since I am dealing with relatively unknown soil material I don't really know how to set the min, max and initial values of flow parameters.
Any thoughts on that?
When the program crashes the following warning is displayed:
"H2D_Clci.exe has generated errors... An error log is being created."
Where is the log saved? What is the name of the log file?
thanks,
Jure

Dear Jure:
It may very well be that your infiltration data do not have sufficient information for a unique inverse solution with many unknown parameters. How many parameters are you trying to estimate from the data? Also, do you have a good estimate of the initial water content? If the soil is initially quite wet, the hydraulic properties in the dry end cannot be accurately estimated. You say that you don't have any idea what soil type you have. Don't you have any textural data (% sand, silt, clay), or perhaps know the textural class (like sandy loam, etc), and bulk density, etc? This would guide you to some reasonable initial parameter estimates. It should also give an estimate of the residual water content using the neural network-based Rosetta (or the Carsel and Parrish catalogue) in the hydraulic properties window. Keeping that residual value fixed may help the inverse solution (make sure that value is less that the initial water content).

Do I understand that you have measure pressure head data in the profile? At what depth? Perhaps try using a free-drainage bottom boundary condition (zero pressure head gradient) at some deeper point in the profile, and use the observed transient pressure head data in the objective funciton to improve the inverse solution (as an observation point). Good luck.--Rien van Genuchten

Hello!
Unfortunately I don't have any pressure head data from within the profile. I also don't have the data on initial head distribution or water content, but as I imagine the soil was quite wet. I only have measurements of water quantity being infiltrated through time at constant head. I am using those flux values as "inverse solution data". It is obvious that with majority of the simulations I'll not be able to get the unique inverse solution.
I have tried using the free drainage boundary but nothing changed since the water front doesn't reach the bottom boundary during the simulation.
I need to estimate all of the soil hydraulic parameters (except l, which is 0,5 in all catalogues anyway). I've also tried to estimate the parameters one at a time. When this manual process seemed to yield correct data I entered those values as initial and set the min and max values accordingly. The calculation process for all 5 parameter at the same time then crashed or did not iterate.
I am dealing with quite heterogeneous material. It consists of relatively high percentage of large pebbles making estimates using Rosetta very inaccurate.
Jure

Jure,
It is well known that infiltration curve can be used to estimate maximum two parameters. Cumulative or actual infiltration curve thus does not contain enough information to estimate more parameters. Therefore, this information is usually used to simply estimate Ks and sorptivity from Phillip analytical solution. When your initial condition was wet, then you have even less information than when the initial condition is dry. There is en infinite combination of sets of parameters that would result in the identical cumulative ifniltration curve, and thus such problem is ill-posed.

Jirka

**ID = 170, High pressure head result**

Dear Hydrus2D mania
I am modelling effect of drain to ground water level.
I used time variable boundary as surface boundary, and no flux at the bottom. For fertical boundary I assigned as a constant flux, freedrainage and constant head for the drain boundary.
It is success to run the model, however when I check the result for pressure head, It gave me value between 2000 cm to -80000 cm while my initial condition is based on pressure head is between -100cm to 650cm (linear distribution with depth).
Any clues for this result, coz I am not sure about this result.
Your help will be valuable for me.

Zal

Zal,
Free drainage boundary condition can not be assigned on sides of the transport domain. It can be assinged only at the bottom. Free drainage BC is actually a unit gradient BC, which means that flux out of the domain is equal to the conductivity for an actual pressure head in a particular boundary node.

Jirka

Dear Jirka,
Thank You for your answer,
I was wrong in my previous information. What I mean with the "FREE DRAINAGE" was "SEEPAGE FACE". Applogies for this mistaken.
Was it make different?
Would you give a comment, why HYDRUS Gave me Higher value of the pressure head (100 fold)? Is it because evaporation is much higher than precipitation?
Thank You
ZAL

Zal,
I do not think that your question is clear enough to me to answer.
Jirka

Dear Jirka,
I am realy sory that i have rised ambiguous question in the forum.
After I check the example of simulation on HYDRUS BOOK (Rassam 2003), my result was ok.
What I meant was, when the soil is very dry, it gave me high negative pressure head. It was strange for me because I have just assigned low initial pressure head. Anyway, I have got the answer now. Thank's
Regards,
Zal

## ID = 171, Deep drainage

Is the groundwater water reference level
for the deep drainage BC wrt the
(0,0) of the drawn finite element mesh, or wrt to
something else (ex., the surface) ?

Hi Ranjan

I beleive that the reference GWL should be with respect to the origin (0,0) of the global axes, and is usually taken as the z-coordinate of the surface.
Sophia

I guess the easiest way to understand this is to see the code. There is a function
real function Fqh(GWL,Aqh,Bqh)
Fqh=-Aqh*exp(Bqh*abs(GWL))
return
end

that calculates the flux. It is called as follows:
Q(n)=-Width(i)*Fqh(hNew(n)-GWL0L,Aqh,Bqh)

where hnew is the pressure in a particular node, GWL0L, Aqh and Bqh are the parameters (entered in the interface), Width is the boundary width associated with given node, and Q is the flux in node.

Jirka

It looks like GWLOL suggests it's wrt to y=0 when the
mesh is drawn. I wonder how the BC flux relates to observation
since the absolute difference between modeled hydraulic head
(which develops from the initial conditions that the
user puts in) and the reference height can be somewhat
arbitrary i.e., does one need to be careful about where
the 0-line goes when defining the mesh ?

I do not think so. If you have your transport domain in different positions (in terms of the z coordinate) then you can just adjust your GWL0L parameter to adjust the function. Your solution does not realy depends on coordinates (z), only on initial and boundary conditions.

J.

## ID = 172, Error detection

Dear all
I was simulating a two-layer soil under atmospheric conditions, and the program was running perfectly. I increased the number of layers (of different soil types) and thereby changed the FEM mesh. The program is not running anymore. It displays the error message before initiating the calculation process.
Does anybody have an idea of what the source of error would be? Is there a somehow systematic procedure to detect the error when the software stops running? like debugging or a certain elimination process, etc.
Is the trial-and-error procedure, the only way out?
Sophia Ghanimeh

Dear Sofia,
Similar with experience.
First simulation was perfect, but when I changed the initial condition, the model is unstable (error). So for the next simulation i try to bring back (import) the initial condition from the first one (the succes one), and it works.
Try and error is not realy bad :-)
Zal

Hi,
Its not "trial and error" as much as matching your mesh with the initial conditions and time step. A coarse mesh requires a different initial time and time step control then a very fine mesh.

When you add new layers to your domain, make sure that you added them by constructing new lines and not by adding new points then connecting. I found building a new domain is better than modifying an old domain as to producing errors.

mazen

## ID = 173, hCritS

Hi!
In the FAQ no. 5 it is stated that surface runoff is initiated immediately after ponding is reached since the hCritS value is set to 0. Does that mean that the pressure head values for the top boundary can not be set above 0 or just that the ammount of water infiltrated is lower than precipitation (due to surface runoff)?
If the latter is true, how can I then simulate positive pressure head at the soil surface and still get the correct flux values?

Jure

Jure,
In Hydrus-2D the hCritS variable is set as default equal to zero and the surface runoff starts immediately after ponding is reached. You can not have positive pressure heads at the soil surface when using atmospheric BC. Hydrus-2D does not consider accumulation of water on the surface.
Things are different in Hydrus-1D. Here hCritS can be specified positive, and the code accumulates any excess water (difference between precipitation and infiltration) at the surface layer until hCritS is reached. Only then the surface runoff starts.

Jirka

Hi Jirka,
Does the non positive pressure head on the surface rule apply to atmospheric BC only or also to constant pressure and variable pressure head BCs?
How can I simulate vertical infiltration from a pool with significant depth of water?
Jure


Jure,
For that purpose you can use either variable or constant pressure head BC.
Jirka


## ID = 174, Flux and velocity

In the boundary.out file:
Q (V/T) is the node flux and v(L/T) is the node velocity. The velocity at a certain node of the boundary surface should be equal to the flux at that node divided by the perimeter of the boundary surface. If those numbers are not matching, where could the error orinFinated from?
Also, if the sum of node flux at the boundary surface is not matching that obtained in WATER BOUNDARY FLUXES OUTPUT, how could that be explained?

mazen


Mazen
The node flux Q (V/T) and the node velocity v(L/T) in the boundary.out must match. v is calculated as Q divided by width associated with the particular boundary node. The boundary width is written in the boundary.in input file (calculated in the interface).

Velocities in the boundary.out file may be slightly different than those in the v.out file. While Q is calculated directly from the discretized Richards equation (described in the manual), v is calculated by applying the Darcy's law, and thus are less precise.

Jirka


## ID = 175, Seed number

Hi,
In order to prevent the use of the same generated field, is there a way to check the seed number generated for scaling factors?

Where could I look up the correlation length x and z and the std deviation in the imput files? Because after runing the file, the parameters of the scaling factors are gone but you can still see the generated field.

mazen

Mazen,
You will not like this answer. You can not check the seed number. I used the standard Fortran routine that generates this number (probably given in Numerical recepies) and do not save the number anywhere. The same is true about the parameters used to generate the random fields. They are not written anywhere in the input or output files. They are remembered in the memory of the Hydrus-2d application, and once you close it, they are immediatelly lost.

Jirka


## ID = 176, Problem with importing initial conditions

Dear HYDRUS users,
I have a problem with importing initial conditions from the previous run results. The new file was created by copying from the previous file using project manager. While the previous file can be run successfully, floating point errors message appeared when execution in the copied file with the imported initial conditions, and the calculation stopped. This problem occurred when I tried to import the inital conditions at any time level besides the first time step.

Is there any solution to this problem without having to rerun the previous simulation again because the simualtion takes a long time?

Preecha


## ID = 177, What is new in Version 2.100?

Hello out there,
have not been here for a while so I noticed Hydrus2D has grown into Version 2.100. Bearing Einstein's "never change a running system" in mind, I wonder if there is any document portraying what has changed since Version 2.080?
Hector


Hi Hector,
all changes are described on the "download page":
http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm -> fill the form -> submit information -> look at comments to versions.
Mirek


Hi Jirka and all others,
I have a question on the registration of the new version of Hydrus-2d. Is the upgrade free for the hydrus-2d users? I understand that there is a new registration method which is different with the old one and therefore the old method is probably not working anymore.

I like the feature with the extended prinitng times (>50) very much as I have done simulations over extended time spans. I would be also pleased if this was incorpurated in a 2.08XX version.

There has been very little rumour about this new version, why is this?

I would be pleased If you could give some more information to me and the rest of the forum.

Greets IJsbrand


Hi IJsbrand,
this version (2.100) is free for existing Hydrus2D users but they must contact IGWMC to get new activation codes.
There are not so many changes in this version and this is the reason why we haven't given it a special attention. All major changes + completely new user interface + new features are planned in version 3.xxx.

I'd like to make several comments to the new licensing system:
1/ New licensing is based on per-computer authorization that uses information about installed hardware to activate/deactivate H2D.
2/ Users who need a network license should wait for next version (2.101) that is supposed to be released in two weeks and that will have an improved support of network licensing.
3/ Why a new licensing? There was a need to improve Hydrus2D protection and therefore we decided to do it. We have had some technical problems with the software for new licensing and currently not everything is as prefect as we expected. However, the new authorization system is fully functional and easy. Our further challenge is to make Hydrus-2D protection as comfortable for our users as possible.
Regards Mirek


Hi Mirek,
when could we expect relaese of 3.xxx?
what major changes could we expect?
regards Uwe

<All major changes + completely new user interface + new features are <planned in version 3.xxx.


Hi Uwe,
we'd like to have version 3.x ready in September 2004 but now it is too early to publish more information about it.

Regards Mirek

**ID = 178, Weighting factors**

Hello,
I tried to calculate the waterflow for a lysimeter. To optimize the saturated hydraulic conductivity I got measuerments of cumulative outflow, watercontent in two depths and suction in 2 depths. Every measuerment got the same weight. In the inverse solution information window I found that the squared residuals of the suction were 10 and 100 times as high as the squared residuals of the watercontent and the cum. outflow. So I controlled the weighting by hydrus because of the variances and the number of measurements for each observationgroup.By calculating the weights in excel (v=1/nb²)I got complete different values (also in relation to each other).
1) Is there a different calculation of weighting factors as described in the manual ?
2) Isn't it better to calculate the weights for the cummulative outflow with the variance of the outflowrates (so the weights become bigger)?

Hi,
Below is the code that calculates weights in Hydrus:
iWeight=1 weighted by mean
iWeight=2 weighted by variance
WT - weight (it is later squared in other part of the code)
I think this should be the same as described in the manual.

Jirka

```
nQ=0
nV=0
nH=0
nTh=0
nC=0
QAve=0.0
vAve=0.0
hAve=0.0
cAve=0.0
ThAve=0.0
do 11 n=1,NOBb
read(5,*,err=902) HO(n),FOS,iType(n),iPos(n),WTS
FO(n)=dble(FOS)
WT(n)=dble(WTS)
if(iType(n).eq.0) then
nQ=nQ+1
QAve=QAve+abs(sngl(FO(n)))
else if (iType(n).eq.1) then
nH=nH+1
hAve=hAve+abs(sngl(FO(n)))
else if (iType(n).eq.2.or.iType(n).eq.5) then
nTh=nTh+1
```

```
ThAve=ThAve+abs(sngl(FO(n)))
else if (iType(n).eq.3) then
nV=nV+1
vAve=vAve+abs(sngl(FO(n)))
else if (iType(n).eq.4) then
nC=nC+1
cAve=cAve+abs(sngl(FO(n)))
11 continue

* ----- Default weighting for observation data ----
if(iWeight.eq.1) then ! by means
Summ=1.
if(nQ+nH+nTh+nK+nV+nC.gt.0)
! Summ=(QAve+hAve+ThAve+vAve+cAve)/(nQ+nH+nTh+nV+nC)
if(nQ.gt.0) then
QAve=QAve/nQ
Summ=amin1(Summ,QAve)
end if
if(nH.gt.0) then
hAve=hAve/nh
Summ=amin1(Summ,hAve)
end if
if(nTh.gt.0) then
ThAve=ThAve/nTh
Summ=amin1(Summ,ThAve)
end if
if(nV.gt.0) then
vAve=vAve/nV
Summ=amin1(Summ,vAve)
end if
if(nC.gt.0) then
cAve=cAve/nC
Summ=amin1(Summ,cAve)
end if
do 12 n=1,NOBb
if(iType(n).eq.0) WT(n)=WT(n)/QAve*Summ
if(iType(n).eq.1) WT(n)=WT(n)/hAve*Summ
if(iType(n).eq.2) WT(n)=WT(n)/ThAve*Summ
if(iType(n).eq.3) WT(n)=WT(n)/vAve*Summ
if(iType(n).eq.4) WT(n)=WT(n)/cAve*Summ
12 continue
else if(iWeight.eq.2) then ! by variance
if(nQ.gt.0) QAve=QAve/nQ
if(nH.gt.0) hAve=hAve/nh
if(nTh.gt.0) ThAve=ThAve/nTh
if(nV.gt.0) vAve=vAve/nV
if(nC.gt.0) cAve=cAve/nC
QSig2=0.
hSig2=0.
ThSig2=0.
```

```
vSig2=0.
cSig2=0.
do 13 n=1,NOBb
if(iType(n).eq.0) QSig2=QSig2+(sngl(dabs(FO(n)))-QAve)**2/nQ
if(iType(n).eq.1) hSig2=hSig2+(sngl(dabs(FO(n)))-hAve)**2/nh
if(iType(n).eq.2) ThSig2=ThSig2+(sngl(FO(n))-ThAve)**2/nTh
if(iType(n).eq.3) vSig2=vSig2+(sngl(dabs(FO(n)))-vAve)**2/nV
if(iType(n).eq.4) cSig2=cSig2+(sngl(dabs(FO(n)))-cAve)**2/nc
13 continue
do 14 n=1,NOBb
if(iType(n).eq.0.and.QSig2.ne.0.)
! WT(n)=WT(n)/(nQ*QSig2)**0.5
if(iType(n).eq.1.and.hSig2.ne.0.)
! WT(n)=WT(n)/(nh*hSig2)**0.5
if((iType(n).eq.2.or.iType(n).eq.5).and.ThSig2.ne.0.)
! WT(n)=WT(n)/(nTh*ThSig2)**0.5
if(iType(n).eq.3.and.vSig2.ne.0.)
! WT(n)=WT(n)/(nV*vSig2)**0.5
if(iType(n).eq.4.and.cSig2.ne.0.)
! WT(n)=WT(n)/(nC*cSig2)**0.5
14 continue
end if
```

### ID = 179, Any experiences whith hydrus2D horizontal plane

Hello again,
I tried for the first time the horizontal plane option in hydrus2d to check an analytic expression. First I applied positive pressures all over the domain effectively representing a confined aquifer. Doing so the hydraulic conductivity parameter is to be converted into a transmissivity. However when dealing with transient cases, the prescription of the storativity (also referred to as the coefficient of storage, a term that describes the volume of water a confined aquifer will release when the water level (sometimes called potentiometric surface) in an aquifer is lowered, remained unclear. It would be convenient to prescribe this value in order to catch its effect on pressure propagation. Any clues how this could be done?

When applying suctions as initial a/o boundary conditions I am in doubt how to interpret the results since it is not clear over which vertical layer thickness the equation is integrated. Any experiences whith hydrus2D horizontal plane

Hector

Hector,
We do not consider "storativity" in the saturated zone, and thus it can not be done. We consider only "hydraulic capacity" for the unsaturated zone.
Jirka

**ID = 180, Particle tracking**

I can't use the flowing particles options in the graphical display of results. I it not yet implemented in Hydrus-2D?

"Moving Particles": This is an option that I have been working on in the past. Since it is not documented in any of our manuals or help files, we have not enabled that in the general package yet. It is going to be an option in the future versions.

Jirka


**ID = 181, Problems with time info. and iteration creterion**

I have been frustrated with the Hydrus 2D simulations of landfill cap percolation rate with clay. I thought the problems were caused by the time discretization and iterating critera input. The values I input are:
Initial time step:0.001; Minimum time step: 0.00001; Maximum time step: 10000. Iteration critera: Maximum number of iterations: 20; water content tolerance: 0.0001; pressure head tolerance:1; Lower limit of the Tension Interval: 1e-008; upper limit of the Tension Interval: 100.
The unit of length is cm, of time is day. When I run the simulation, the cumulative flux across the lower boundary of landfill cap jumps from $10^{-1}$ to $10^{12}$.
Would you please give me some hints what might be wrong? Thanks

Xiaoli Liu


I would suggest that you use the van Genuchten-Mualem model with 2 cm air entry value for the soil hydraulic properties. That typically stabilizes the numerical solution, while at the same time being better in describing the hydraulic conductivity for heavy textured soils.

Jirka

**ID = 182, How to set time info and iterat. criteria for clay**

I am still in trouble with the simulations of clay. Hydrus 2D works well if I chose silt. If I use clay, however, I saw stars under hatm, hroot, hdrain, and it runs extremly slow. If I chose silt and then change Ks to 2(Ks = 2cm/day) without changing anything else, it works, but the results don't make sense. The cumulative flux is higher than that if I use silt (Ks = 6cm/day). Would anybody give me some advice to solve this problem?
Thanks
Xiaoli Liu


Hello,
Did you try changing the mesh size. Try both a very fine mesh size and a coarse one, and see what the difference would be in runing the simulation. If you are not doing

transient state and you are only interested in steady state conditions, you can chose silt characteristics with the conductivity of clay.

You can always try starting with a much lower initial time step.
Thanks,
mazen

Hi Xiaoli Liu,
I am a little confused about your question but i have a suggestion for you. Double check your initial conditions, plot your hydraulic conductivities of your scenarios as a function of the pressure head and compare your initial conditions and simulation pressures with the hydraulic conductivity. This might give you more insight into how much cumulative flux you expect relative to different scenarios and under what conditions you are simulating the flow.

Defne

Xiaoli,
I recommend you, when simulating flow in clay, to use the van Genuchten-Mualem model for the soil hydraulic properties with the air entry value of -2 cm. The original van Genuchten-Mualem model is often not adequate for clays and it predicts very fast decrease of the conductivity with the pressure head. The model with the air-entry value of -2 cm predicts much better the unsaturated soil hydraulic conductivity, and also makes model (Hydrus) much more stable.
Jirka

Jirka is absolutely right. When n is less than about 1.1 (probably for the clay soil), use the option with the air entry value (minimum capillary length) set to -2 cm. Good luck.--Rien.

## ID = 183, Dual porosity

Hello,
I am doing inverse optimization of combined tension-infiltrometer-tensiometer measurements in a macroporous soil. I would like to use Hydrus-2D and a dual porosity model. In the version of hydrus2d i have, the dual porosity model is 'visible' in the graphical interface but not available. Is this option already available in another version of Hydrus-2D and would it be possible to use it? If not, is there another way to somehow take macroporosity into account in Hydrus-2d?

Thank you
Sofie Herman

Sophie,
Such version is under development at present, but not available yet.

Jirka

In lieu of dual porosity, does Hydrus1D or Hydrus2D simulate preferential flow by any other method (such as the Addiscott mobile-immobile capacity model)? Thanks.
Tom Nolan

Tom,
Both Hydrus-1D and Hydrus-2D can simulate accelerated solute transport using the concept of mobile-immobile water. See the manual.
Jirka

In lieu of a dual porosity model in the current release of HYDRUS-2D, I wonder how much loss of accuracy would result from simulating water flow in a daily irrigation regime for an agricultural crop or pasture on a well-structured clay loam to light clay (oxisol). The area to be modeled is a warm sub-tropical region with high summer rainfall, biologically active topsoil resulting in extensive network of macropores. I wonder also about loss of accuracy in water flow modeling for a vertisol (common in Australia) with high proportion of shrink-swell clays.

Tony McCardell

Further to my last message. Should one use actual Ksat or "matrix" Ksat (Ksat through the soil matrix only, excluding the macropores) when modeling flow in a macroporous soil? I ask this also because I understand some methods of measuring Ksat, at a slightly negative pressure, aim to eliminate the contribution to flow due to macropores.

Tony McCardell

Tony,
I would say that water flow in a soil with in extensive network of macropores (with high proportion of shrink-swell clays) is not describe well with the Richards equation. Since Hydrus-2D is based on the numerical solution of the Richards equation, this model may not be appropriate for your situation, and some kind of description of the preferential flow may be necessary.

I'm currently developing a version of Hydrus-2D that will have several options that will try to approximate preferential flow. See my paper (that deals only with 1D):

Simunek, J., N. J. Jarvis, M. Th. van Genuchten, and A. Gärdenäs, Nonequilibrium and preferential flow and transport in the vadose zone: review and case study, Journal of Hydrology, 272, 14-35, 2003.

Look at this paper and let me know, which option you think would be best suitable for your particular problem

Jirka.

Dear Jirka
I read your article as you suggested, with interest...
Simunek, J., N. J. Jarvis, M. Th. van Genuchten, and A. Gärdenäs, Nonequilibrium and preferential flow and transport in the vadose zone: review and case study, Journal of Hydrology, 272, 14-35, 2003.

In spite of the fact that my soil is macroporous, I'm proceding with HYDRUS because the macropores seem to be mostly in the top 30-60cm. And, yes, the soil is shrink-swell to some degree (as I found out when I dried some samples), but not nearly so much as some other soils in the area. It is also irrigated every day so that the soil moisture content is nearly always towards the wet end of the soil moisture curve. So I'm going to use HYDRUS as planned, and keep my fingers crossed.
Tony

**ID = 184, No access**
Access not allowed to this forum

**ID = 185, Hydrus-1D in public domain!**

It has been decided to release Hydrus-1D and STANMOD as public domain programs. You can download their latest versions at
http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm

**ID = 186, STANMOD in public domain!**

It has been decided to release Hydrus-1D and STANMOD as public domain programs. You can download their latest versions at
http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm
**ID = 187, No access**
Access not allowed to this forum

**ID = 188, No access**
Access not allowed to this forum

**ID = 189, Thanks for your help!**

i am a freshman to use the hydrus1d, so, is there anybody who once used the hydrus software to deal with the water and salt transport simulation? please tell my your E-mail, i have some questions to ask you for help! thanks a lot! best regards!

Hi,

there are almost 1000 customers who use Hydrus1D and Hydrus2D for such simulations. This discussion forum has been created to help people who have some problems/questions regarding Hydrus1D - you can post your questions here. But, as this service is provided for free, we can't guarantee that all questions will be answered and/or how long time it will take. Please visit http://www.pc-progress.cz/Fr_Services_Hydrus.htm for information about other available services.

Regards Mirek


**ID = 190, Furrow irrigation**

Dear Jirka,
I am trying to simulate water and solute flow in a furrow irrigation system. I chose time variable BC and number of time variable records is 5 and set different ponding depths for these times in GWL. The 5th row under the time (48 h) column is set to 0 cm ponding depth. I run for 96 hrs. It means that from 48h onwards ponding is 0 but in the cumulative boundary flux out I still see the water being pplied.

How do I set that? What I mean is there shouldn't be water infiltrating after the time that is set to 0 cm GWL.

thanks
Thabo


Thabo,
Water can infiltrate both at positive and negative tension. Thus infiltration does not stop when the pressure in the furrow is equal to zero. At the moment when there is no water in the furrow, you need to change boundary condition from specified pressure to specified flux (zero). You can do that in two ways:
a) Run two simulations. The first one with pressure BC. The second with flux BC, while you import the final pressure head profiles from the first run as initial condition for the second.
b) In some version of Hydrus-2D it was possible to specify GWL.gt.999999 and the code interpreted that az zero flux and switched BC automatically. Try that but it may not work. If it does not send me email, and I will send you version where this is active.

Good luck
Jirka


Dear Jirka,
It works when I run two times but I would like to have the version that does it automatically.

Under the field condn, water is pumped from the inlet of the furrow and the ponding happens depending on the hydraulic properties of soil and the pumping rate. But in HYDRUS I force the soil to have different pressue heads with time and ran the model.

1. Is there a way to get the ponding when I apply the variable flus boundary from the inlet? and the ponding depth to get back slowly to 0 after irrigation?
2. Now, Can I set the furrow to be in variable head boundary and the row to be in the atmospheric?
3. How do I set the root zone to be in the row? when I set the root it always comes in the furrow?
Thanks
Thabo


Thabo,
I sent you the code.
a) No. You need to specify variable pressure head with time. The code will not automatically decrease head due to infiltration.
b)Yes
c) I do not see any reason why you shouldn't be able to have roots only in furrow. Remember, however, that potential traspiration is associated with atmospheric BC. This means that PotT is multiplied by length of the atmospheric BC, and this volume is distributed over the root zone. If you want to associate T with different length, you need to manually rewrite the value in the atmosph.in file.

Jirka


### ID = 191, HYDRUS-RD Manual

Dear
I want to but HYDRUS-2D software, but before purchesing I want to check manuals of Hydrus-2d in detail to see is it proper for my PhD research or no.
Any body can send me PDF files of manuals?
Regards,
Behzad


Hi Behzad,
you can download the manual from http://www.ussl.ars.usda.gov/pls/caliche/tftpinfo?model_in=hydrus2d or http://www.ussl.ars.usda.gov/models/hydrus2d.HTM.
But sometimes it is not possible to download documents from this site because of some technical problems (I tested it today...). Therefore I'm going to post the H2D manual also to our WEB site (http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm). The manual should be available for downloading tomorrow.
Mirek


### ID = 192, Variable flux B.C.

Hi HYDRUSers,

What happens if the variable flux that i force into the system is more than it can take in? I know that with atmospheric B.C., HYDRUS will allow head to build up to a value set by hCritS and then remove the water that can't infiltrate and we call that water runoff.

I used variable flux for my rain input and noticed positive heads forming at the boundary. Is it true that variable flux boundary condition will allow head to build up so that the flux forced into the system can actually go in? Is there a way to specify the limit for this head buildup? Alternatively, is there a trick to allow no head to build up (i.e. all positive pressure to become runoff just like it does in atmospheric B.C.) with variable flux B.C.?

The reason for my question is that, to model crack infiltration, i am forcing more water into the crack (all that falls upslope of the crack) than it rains on top of it. Then, to model infiltration on the sides, i want to input rain just as it rains. In other words, if possible, i want to try inputing rain at different intensities at different locations of my mesh.

Defne

Defne,
There are two types of boundary conditions in HYDRUS: a) system independent, i.e., those that are inforced no matter what (such as constant and variable head or flux), and b) system dependent, i.e., those that take into account status of the system and adjust accordingly (such as atmospheric BC, seepage face BC, Free drainage)
Constant and variable head BCs belong to the first group and thus are enforced at any cost, even it it means to increase the pressure at the boundary above saturation.

I have a version of Hydrus, that treats variable head BC, the same way as Atmospheric BC. I will send you this version.

Jirka

Hi All,
What happens when precipitation, evaporation, transpiration (for plant water uptake), and irrigation are simulated simultaneously? What is the proper way of entering both atmospheric boundary conditions and irrigation data so that the program recognizes these inputs and treats them differently?

I am trying to simulate a flow problem for one year including both boundary types at the top boundary. So far, I handled irrigation inputs under atmospheric boundary condition. I added irrigation amount to precipitation. I did not know any other way of handling both atmospheric boundary conditions and variable boundary flux (in my case, irrigation), so I choose this way intuitively since both precipitation and irrigation are inputs in the mass balance. With Jirka's explanation below, it is clear that both boundary conditions are treated differently by the program. I am not sure now if my way is appropriate? Since I am treating irrigation as atmospheric BC under precipitation, I have been having convergence problems during irrigation events when

the total precipitation input (i.e., precipitation + irrigation, or only irrigation during summer) exceeds the infiltration capacity of the top soil. I kept seeing "floating point error- Overflow!" error message. In one of the recent runs, I forgot to check "Root Water Uptake" even though I had all inputs (precipitation +irrigation, evaporation, and transpiration) entered under "Variable Boundary Conditions" window. Interestingly enough, that is the only simulation that runs for the entire year with no convergence problem. I checked "Root Water Uptake" option and re-ran the same problem with the same inputs. But the program stopped giving me the same floating point error message.

Does anybody have similar problems? Besides changing domain discretization, what other alternatives do I have to tackle floating point error? And what is the proper way of entering both atmospheric and variable boundary conditions at the top boundary?

Sevim

## ID = 193, Modeling precipitation and dissolution

Is it somehow possible (besides modeling non-equilibrium sorption) to model the precipitation and dissolution of substances with Hydrus-2D?

The only way to do that is to use the kinetic sorption option. You can assume instead of sorption, precipitation/dissolution, and use the same concept (equations).

Jirka

If you mean precipitation and dissolution of salt (say
calcite, or halite), I would be curious to know what types
of ranges can be used for the transport parameters in hydrus.
(I have a reference for Na+ ions).

No, i'm trying to model precipitation and dissolution of PO4. I'm already using the adsorption parameters for two-site sorption of PO4. I guess I will have to wait for the coupled Hydrus-Phreeqc model. How is the coupling going, Jirka?

I have the coupling done to the point that I can simulate reactions such as cation exchange, complexation, precipitation/dissolution, both kinetic and instantaneous. But I can not still do redox reactions, surface complexation.

J.

## ID = 194, Athmosperic boundary condition with surface layer

Hi,

I have some strange results (at least they seem so to me) using HYDRUS 1D and the atmosperic boundary condition with surface layer. The 'help' says that this boundary condition permits water to buid up on the surface.

However, in my case it does not seem to do this. I have a 2 m deep profile, 401 nodes and let it rain 19.2 mm/day for 5 days and then let the profile drain (O rain)... The results I get looks like this after 10 days (in mm):

sum(rTop)sum(vTop)sum(vBot)hTop hBot
-19.2 -8.89 -2.12 0 -100
-38.4 -17.4 -4.24 0 -100
-57.6 -25.9 -6.36 0 -100
-76.7 -34.4 -11.5 0 -0.159
-95.9 -42.9 -19.5 0 -0.154
-95.9 -42.9 -25.8 -187 -11.8
-95.9 -42.9 -30.8 -237 -22.6
-95.9 -42.9 -34.9 -272 -28.7
-95.9 -42.9 -38.8 -300 -35.8
-95.9 -42.9 -42.3 -323 -43.6

What I don't understand is that h never goes positive? Where does the water that does not infiltrate go to ? Actually changing boundary condition to atmospheric with surface runoff does not change the simulation..

Does anybody have any experience with this boundary condition or sees what I am doing wrong?
Jan


Jan,
When you select "atmospheric BC with Surface Layer" then go to the atmosph.in file and modify manually the hCritS as below to whatever you want the maximum water layer to be.

Jirka

*** BLOCK I: ATMOSPHERIC INFORMATION **************************
MaxAL (MaxAL = number of atmospheric data-records)
183
hCritS (max. allowed pressure head at the soil surface)
1
tAtm Prec rSoil rRoot hCritA ht


**ID = 195, Constant head BC changes!**

Hi,
I assigned a constant head BC = 0 at the bottom of the domain. As initial pressure head, I also assigned zero at the bottom nodes. When I check H.TXT file for nodal

pressure heads at print times, I see very small negative head values, like -5.5*E-25, at each print time for the bottom nodes where I assigned constant zero head. Constant zero head must remain the same during the entire simulation. I have no clue why this is happening. Does anybody have any explanation for this?

Sevim

Sevim,
How big do you think is the difference between zero and -5.5*E-25?
It is exactly -0.000 000 000 000 000 000 000 000 55.
Do you think that's something to warry about?

Jirka

**ID = 196, Heat transport**

Jirka and Mirek,
Do you have some examples on the coupled transport of liquid and heat?
Please let me know..

Thanks,
mazen

Mazen
What type of examples are you looking for?
From examples distributed with our software, Test7 of Hydrus-2D (infiltration of warmer water) and Test2 of Hydrus-1D include heat transport.

I have recently worked on several aspects of heat transport.

1) Heat transport in the systen with heat pulse probe
Hopmans, J. W., J. Šimùnek, and K. L. Bristow, Indirect estimation of soil thermal properties and water flux from heat pulse measurements: Geometry and Dispersion effects, Water Resour. Res., 38(1), 10.1029/2000WR000071, 7.1-7.14, 2002

2) Coupled heat, vapor and water transport:
Scanlon, B., K. Keese, R. C. Reedy, J. Šimùnek, and B. Andraski, Variations in flow and transport in thick desert vadose zones in response to paleoclimatic forcing (0 – 90 kyr): Monitoring, modeling, and uncertainties, Water Resour. Res. 39(7), 1179, doi:10.1029/2002WR001604, 13.1-13.7, 2003.

3) Freezing and thawing:
76. Hansson, K., J. Šimùnek, M. Mizoguchi, L.-Ch. Lundin, Water flow and heat transport in frozen soil: Numerical solution and freeze/thaw applications, Vadoze Zone Journal. (submitted August, 2003, accepted).

Jirka

**ID = 197, Authorization for STANMOD-2D**

We have found out that STANDMOD installation program was not updated correctly on our WEB server. Therefore, if you downloaded STANMOD before January 10, you don't have authorization to work with two-dimensional projects. The installation program (with fixed authorization) was updated today.

**ID = 198, Internal drainage experiment**

Hi,
I just tried some simulations with Hydrus 1D for a simple internal drainage experiment, starting from a 100-cm saturated profile (initial condition h=0 cm) and the soil surface covered with a plastic sheet throughout the experiment (upper BC: constant flux q=0 cm/day). Below I have free drainage conditions. The model only runs when I select 'seepage face' as a BBC, but then of course gives a constant h=0 at 100 cm depth throughout the experiment whihc is not realistic. If I select the free drainage BBC, I got a message 'not enough data'. I played a bit around with the iteration criteria and profile discretization, but it doesn't work out. Any help?

Wim

Wim,
I have searched the code and have not find any possible message 'not enough data'. Thus I can not really say what your problem is. Feel free to send me the zipped project (folder + Project_name.h1d) to Jiri.Simunek@ucr.edu.

Jirka

**ID = 198, Internal drainage experiment**

Wcorneli, 02/13/2004 : 17:31:06
Hi,
I just tried some simulations with Hydrus 1D for a simple internal drainage experiment, starting from a 100-cm saturated profile (initial condition h=0 cm) and the soil surface covered with a plastic sheet throughout the experiment (upper BC: constant flux q=0 cm/day). Below I have free drainage conditions. The model only runs when I select 'seepage face' as a BBC, but then of course gives a constant h=0 at 100 cm depth throughout the experiment whihc is not realistic. If I select the free drainage BBC, I got a message 'not enough data'. I played a bit around with the iteration criteria and profile discretization, but it doesn't work out. Any help?

Thanks for your help.
Wim

Jirka 02/14/2004 : 00:41:46
Wim,
I have searched the code and have not find any possible message 'not enough data'. Thus I can not really say what your problem is. Feel free to send me the zipped project (folder + Project_name.h1d) to Jiri.Simunek@ucr.edu.

Jirka

http://envisci.ucr.edu/faculty/jsimunek/default.htm

## ID = 199, Rainfall input

I am experimenting with test 1
I wish to simulate a constant rainfall of 10 mm h-1
do i add this through the time variable boundry conditions by adding one row of data
or do i add this through the constant flux boundary conditions ? i can find no units for this ?

also, i have added 3 observation nodes, when i view the graphs i see 3 lines but no indication of which colour relates to which node ?
clearly these are straightforward but not obviously apparent to us, any help much appreciated

Dear ecmiller72,
Anonimity: First, It would be fair if you register in our discussion forum under a full name and affiliation (that's a message to others as well). You expect a free help with your problems and thus you should not be anonymus.

Boundary conditions: You can specify a constant flux at the soil surface both ways. You can either specify a constant flux boundary condition and in the next window give the flux value (remember - Infiltration is negative!!!) or you can select atmospheric BC. In the latter case you need to specify that you have time variable BC with one record and specify the flux in the column Prec.

Units: The units must be consistent with the rest of the project. Units are selected in Time and Geometry information dialog windows.

Observation nodes: Click on the graph with the right mouse button. From the hanging menu select "Legend" and select "Visible". You will get information about which color correspond with which observation node.

Jirka (non-anonymus)

## ID = 200, MSO experiment

I am having difficulty with the Hydrus 1-D inverse procedure for calculating hydraulic parameters form multi-step outflow (MSO) data. I think I have likely made an error in specifying and coding the "Variable Boundary Conditions" or the "Data for Inverse Solutions".

I have a large data set (2685 points) from a MSO experiment (sample height = 5 cm) on an undisturbed soil (Silt Loam). The sample is saturated to start (Theta = 0.38) and then three pneumatic pressure steps are imposed (100, 250 and 500 cm) from the top of the sample. The bottom is open to air pressure and water is collected in a burette.

"Time Variable Boundary Conditions", I have inputted the Time and the imposed pneumatic pressure at each time, under "hTop", as follows.

Time | hTop

30 | 100
:
(data every 30 s)
:
82050 | 500

The experiment is run from 0 to 82050 seconds, with transducer measurements taken every 30 seconds. Measurements of matric potential (h(t)) are taken from a tensiometer inserted 2 cm into the sample. Measurements of volumetric water content (theta(t)) were calculated from outflow measurements, bulk density and saturated water content.

"Data for Inverse Solutions" is inputted in columns as follows

X | Y | Type | Position | Weight
30 | -61.1431 | 1 | 1 | 1
:
(data every 30 s)
:
82050 | -405.685 | 1 | 1 | 1
30 | 0.347316 | 2 | 0 | 1
:
(data every 30 s)
:
82050 | 0.155699 | 2 | 0 | 1

I have selected to specify water content as the "Initial Condition" and have chosen 0.38 for all depths in the initial profile.

If I choose "Silt Loam" in the "Soil Hydraulic Parameters" section, the numerical solution does not converge, but if I choose "Sandy Clay Loam", I am able to get results. However, in the "Observation Points", theta(t) is just a straight line at 0.18, which tells me that I have not coded that input data correctly. In the "Inverse Solution" section, it has stopped after 2 iterations and the WCS = 0.1885, which is clearly too low. Again, indicating that I have not coded the theta(t) data correctly.

If I try to specify the saturated water content or saturated hydraulic conductivity (both of which have been measured independently), an error log is created and I am instructed to restart the program.

Any ideas on where I am going wrong?


It seems that you are doing most of the things right. I would do the following:
a) Specify initial condition in terms of the pressure head.
b) Let at least one node (at the top) to be unsaturated (e.g., -0.1 cm)
c) Check your input with input in our example problem Multflow in Inverse workspace.
d) If nothing works, zip particular project and email it to me.

Jirka


### ID = 201, Print a list of all simulations + descriptions

dear all,
Does anyone know if it is possible to copy a list of all simulations and the description of the simulations.
It is nicely listed in the project manager [thanks codecrackers]. But I wanted to tell the world what simulations I did and I retyped a part of them manually (a lot of work). I used also a screen dump but this does not look very proffesionally. The list of the names of the simulations and there describtion is stored in some file but which and can I read as a text file ( then my problems are solved.)
When I can get the list I can change my screen dumps (ya, more than 6) because the windowsize can be altered .

I hope on a soon reply
With kind regards
IJsbrand

IJsbrand,
I have not been getting notifications last month about entries into discussion forum. That's why I have not been answering. Sorry.

There is only a list of workspaces in the hydrus2d.ini file in the "WIndows" folder. Then Hydrus knows where to look for Hydrus projects. There is no list of projects. That list is dynamically created when Hydrus searches particular (active) workspace from project_name.h2d files and descript.txt files (in project folders).

Jirka


### ID = 203, Listing of simulations

Dear all,

Does anyone know if it is possible to copy a list of all simulations and the description of the simulations.

It is nicely listed in the project manager [thanks codecrackers]. But I wanted to tell the world what simulations I did and I retyped a part of them manually (a lot of work). I used also a screen dump but this does not look very professionally. The list of the names of the simulations and there description is stored in some file but which and can I read as a text file (then my problems are solved.)

When I can get the list I can change my screen dumps (ya, more than 6) because the windowsize can be altered.

IJsbrand

IJsbrand,

The descriptions are in descript.txt for each simulation. Unless you have hundreds of simulations you can probably open each one and copy the text. Alternatively, if you numbered your simulations, you can write a simple code that will open each file and copy the text within to a place you like, close the text file and move onto the next simulation description. You can also do this if your simulations are not numbered consecutively: you would have to write a short code that will open each file based on the simulation name and copy the inputs to a place you like. This is how i would do it.

Defne

## ID = 205 Version incompatibilities

Hi!

I've created several simulations using Hydrus2D 2.101. Due to some unrelated reasons I have to regularly change the hardware configuration of my computer lately. This causes Hydrus to turn into demo version.

I've installed my old 2.02 version of Hydrus and realised it does not recognise some of the files created with the new version. It reports: "unexpected file format" & "error reading file meshgen2.pmg"

How can I use the new simulations with the old version of Hydrus?

Jure

Jure,

Hydrus versions are only forward compatible (like most softwares). That means that new versions can read input files from older version, and older version can not read input files from newer versions. This incompactibility is only in the meshgen module, since other input files were not changed in 2.0x subversions.

Recent versions (from 1-1-2004) have software protection (which I personally was against implementing, but was over-voted by other members of the Hydrus team). If you need to change computer, I believe that you need to get a new authorization from IGWMC (free of charge).

Jirka

**ID = 206, Version incompatibilities**

jkrivic 03/16/2004 : 13:05:59
Hi!
I've created several simulations using Hydrus2D 2.101. Due to some unrelated reasons I have to regularly change the hardware configuration of my computer lately. This causes Hydrus to turn into demo version.
I've installed my old 2.02 version of Hydrus and realised it does not recognise some of the files created with the new version. It reports: "unexpected file format" & "error reading file meshgen2.pmg"
How can I use the new simulations with the old version of Hydrus?

Thanks,
Jure


jkrivic 03/18/2004 : 10:59:22
The 2.02 version (as stated on the CD sticker) turned out to be 2.05 version.
Furthermore, I've installed the 2.007 version and it also doesn't work.
An advise from tech support would be appreciated.

Thanks, Jure


**ID = 207, Solute transport BC**

Hi,
I'm new in Hydrus2D modeling. I do not understand the method of defining solute boundary conditions: when I choose for example the "first type" botton in the BC window, and select nodes, a message appears "pointer to the vector of boundary conditions". What is that mean ? how do I specify the concentration assigned to the nodes ?
Hila

See FAQ 29.
J.


**ID = 208, Infiltration of water into soil column**

I was wondering if anyone could help me determine if my results make sense. I am trying to simulate infiltration of water into a hypothetical 6 inch soil cover in an arid region. I have precipitation and evaporation data that I have entered into the model. I have used Atmospheric BC with surface layer and a free drainage lower boundary condition. My results have negative inflows, small positive top fluxes and negative bottom fluxes. Does this mean that I have no water infiltrating into the bottom of my soil profile? I also have a high WatBalR ranging between .234 and 8.4. Is this a problem?

Thanks For your help

a) Hydrus-1D assumes that the vertical z-axis is positive upward. That means that positive fluxes are upward (evaporation at the soil surface) and negative fluxes are downward (infiltration at the soil surface or drainage at the bottom of the soil profile).

b) Water mass balance error (relative) WatBalR should be less than 1%. You need to either use stricter iteration criteria (water content tolerance < 0.001) or finer discretization.

J.

Hello:
I am using hydrus 1D to run infiltration simulation through a clay column of 100 cm length. The initial condition is an initial set of pressure heads (negative as it is unsaturated zone).The boundary conditions with 0 pressure head t the bottom face due to a water table and a constant rate of infiltration of -0.9cm/hr (negative sign convention used). I am using all other default parameters and am using the option of the -2cm air entry value.
The problem is that my output shows the pressure head variation going to positive values in the vadose zone. This is not possible, as vadose zone is known to have negative head values. Could you suggest where my input is lacking?
Thanks

Perhaps your saturated hydraulic conductivity is smaller than specified infiltration rate (0.9cm/hr). Then obviously you need to create a positive gradient, since the pressure at the bottom is fixed to zero.

Jirka

Yes, you are right, increasing the hydraulic conductivity to greater than the infiltration does give desired results.
However, when I change the initial condition to water content instead of pressure head, why does'nt the solution converge:(? Coz, I am using the negative air entry option, my infiltration is -ve and saturated hydraulic conductivity is more than the infiltration?
Thanks

Perhaps your saturated hydraulic conductivity is smaller than specified infiltration rate (0.9cm/hr). Then obviously you need to create a positive gradient, since the pressure at the bottom is fixed to zero.

Jirka

Jirka,

I am trying to compare simulation results of Hydrus 1-D with that of the analytical solution of Srivastava and Yeh 1991, for a clay comuln of 100 cm length, steady infiltration of 9 cm/day and preence of water table at the lower bottom of the column. I also have provided initial condition in form of water content.However, what steps do I need to adopt in hydrus-1d to get esults as close as possible to the analytical solution?

Why is it that when using hydrus 1 d to simulate steady state infiltration through a 100 cm clay column, the output shows that at times greater than 0 , the water content near surface is very close to the saturation value? This would be the case only when there was ponding at the surface and that is not what I put in the input.
Any suggestions as to why this could be happening?

The saturation or pressure head at the surface will be very close to values corresponding to the hydraulic conductivity = applied flux.
J.

Is there a way..the soil summary editor can be exported to an excel file, or an excel file can be exported into the soil summary editor?
Also....if I open the output files with an excel program...the formatting is lost...any suggestions how I cud prevent that?

1) You can just select the data in the table and click Ctrl+C to copy the content into the clipboard, and then in Excel click Ctrl+P to paste. The same works in both direction, i.e., Hydrus to Excel and from Excel to Hydrus.
2) This really deals with Excel. Excel ask you whether the data are delimited or not.

Jirka

I am trying to simulate the wetting profile in a 100 cm clay column with constant infiltration of 0.9 cm/s(-ve sign convention), saturated hydraulic conductivity of 1 cm/s (+). The initial condition is a set of water content values.I am using the -2cm air entry value option but no hysterisis.The max water content is 0.4 and min is 0.068.
I have obeserved in the output water content profile for times greater than 0 shows that water content near the surface is very close to the saturation water content (max). However when I compare the results to another analytical model, and the analytical model shows the water content profile near the surface to be gradually increasing from min to max. I am aware that my infiltration is almost equal to the saturated hydraulic conductivity...but then why is the analytical solution different from the Hydrus1d result and which one ia the more correct representation?

Hello:
I was wondering if hydrus1D takes into account preferential flow or not? As per my understanding, it does not, but I wished to double confirm.
Thanks

The standard version has only option to use the physical nonequilibrium using mobile-immobile water concept. We are working on a new version that will have several option on handling preferential flow (different dual-porosity and dual-permeability approaches based on my paper in Journal of Hydrology in 2003). We plan to release that version sometime next summer.

Jirka

Does the option of using -ve air entry value with Mauelm's equation take into consideration the option to use the physical nonequilibrium using mobile-immobile water concept?
In the profile output, I see plaots of 5 different colors corresponding to the 5 times that I gave in the input print file. However, when I go to the plot and left click "series" it shows 6 differemt times(T0,T1,T2,T3,T4,T5)...why is this happening?
And how can I know what plot corresponds to what time?

### ID = 209, Solute transport: output units

Hallo Hydrus users,
I tried to calculate the transport of oxygen in a soil column and the decay of oxygen because of biological activity. The transport as diffusion in gaseous phase works fine, and the results are plausibel. There are any restrictions, for example the model does not calculate the gas convection - this can lead to high concentrations during wet time intervals. The decay rate I defined for the upper soil material (SinkG1). The calculation works, but I have problems to understand the unit of output (cumulative first order reaction in "solute.out"). What does "ML-1" or "grams per centimeter" means? I used a vertical plane of the soil column.

Ralf

Ralf,
"Cumulative first order reaction" is integral in time of the first order reaction over the entire domain, i.e.,

S= Integral over t { integral over domain [Sinkg1*C_g*a]*dA*dt,

where
SinkG1 - first order decay [1/T]
c_g - concentration in the gas phase [M/L3]
a - air volumetric content [-]
dA - domain [L2]
dt - time [T]

S = [1/T]*[M/L3]*[-]*[L2]*[T]=[M/L]

You can interpret that as decayed Mass in the transport domain per unit length in the third dimension.

Jirka


## ID = 210, Output units for concentration

Hi,
I am a fairly new user of Hydrus.
I am using the " export all " feature in the Graphical Display of Results window to get all output data for a single cross-section. When I do that, I get a text file with various columns (W1,W2 etc.) which I understand are Pressure head, Water Content etc. In what units are the concentrations written to this file ? If I entered mg as units for mass and cm for units of length then would these be in mg/cc ?

Thanks,
Sheena


Sheena,
This is correct. Units are the same as used throughout the simulation and as selected by user. Concentrations are in [M/L3], thus in your case [mg/cm3].
See also FAQ 31.

Jirka


## ID = 211, Constant pressure

Dear All,
I would like to simulate the groundwater fluctuations as the effect of drains. I am still not so sure what the Vertical boundary should be, should I assign on the vertical boundaries as Constant Pressure, or is that enough if I assigned Initial Condition in Pressure Head and leave the vertical boundaries as No Flux?? or Can I assign both in my simulations?
Zal


Zal,
If you can assume that there is no vertical inflow into the domain, you should use zero flux BC. If, however, you believe that there is a horizontal flow, such as when the domain is in the hillslope, then you are in trouble, since there is no good BC to describe that.

I have been looking at the option to use "specified gradient BC", which would keep the gradient constant, equal to the hillslope. This is not implemented in the Standard Hydrus, but I could send you that file, if you want to use that option.

J.

Dear Jirka,
Even though my domain is not in the hill slope, but it has hydraulic gradient and there is horizontal flow occur. That's why I can't receive the best result.
I will be grateful if you can send me that file, because I have to run this simulation and I cannot change the boundary.
Zal


Zal,
I have emailed you the updated program. Below is the description of this new option.
Jirka

Gradient-type boundary condition

At present Hydrus-2D implements gradient type boundary condition only as Free drainage, or unit gradient BC. However, in many situations one also needs a non-unit gradient BC. For example for flow in a hill slope where the gradient is parallel with the direction of the slope. I have recently implemented such option, that on boundaries with specified "Free drainage" BC applies specified gradient. Gradient is positive for flow from right to left, and should be used only on sides of the transport domain.


## ID = 213, MESHGEN / polygon and a circle troubles

Hello out there,
I am experiencing some troubles in building meshes consisting of polygon and a circle. I want to build a mesh to represent a series of horizontal wells (vertical plane) and taking symmetry into account the domain consists of a rectangle with a well onone side. The well is considered by a an arc.
Every thing works out fine, when the domain size relative to the arc is small. But as soon as I increase the domain size (length 16 m, width 3 m, diameter of circle 0.1 m), the boundary shape of the arc is ignored and the domain is closed between the 2 edge points of the arc (on the symmetry axis).

Surprisingly this bug seems not to occur each time.
Has anybody some clue to muddle through this strange behaviour?
Regards
Hector


I found the solution for the troubles I encountered:
One has to adapt mesh density at the circle's boundary points, so that an appropriate spatial resolution is achieved on the curved boundary.

Hector

**ID = 214, TROUBLES with polygon and a circle**

Hello out there,
I am experiencing some troubles in building meshes consisting of polygon and a circle. I want to build a mesh to represent a series of horizontal wells (vertical plane) and taking symmetry into account the domain consists of a rectangle with a well onone side. The well is considered by a an arc.
Every thing works out fine, when the domain size relative to the arc is small. But as soon as I increase the domain size (length 16 m, width 3 m, diameter of circle 0.1 m), the boundary shape of the arc is ignored and the domain is closed between the 2 edge points of the arc (on the symmetry axis).

Surprisingly this bug seems not to occur each time.
Has anybody some clue to muddle through this strange behaviour?
Regards
Hector

**ID = 215, Problems with automated e-mail notifications**

4 weeks ago our IS provider changed some components on our WEB site server and since that time automated e-mail notifications (about new topics, etc.) didn't work. Also after inserting a new topic or reply an error message was displayed although the topic was inserted correctly. We appologize for these technical problems, now it should be working correctly.

**ID = 217, Snow**

Dear All,
I have three questions concerning Hydrus-2D:

1. I want to model with variable climate conditions which are characterised by high snowfall events during winter. Are there any plans in future to include and to implement the option that would take care of snowfall-snowmelt events ?

2. When i start the flow animation of my graphical results without the option "Color Spectrum for All Time Levels", Hydrus-2D always automatically activates this option again so that i only can use the "one level forward" buttom for changing the time levels. is there any solution for this problem ?

3.Is it possible to change the color pattern scale for the graphical display to an unregular pattern ?(for example a pattern that contains one color for all matrix potentials between -10 up to -200 and a regular scale pattern for all other matrix potentials)

Thanks
Marc

Marc,

1. I have been looking at the following simplified description of snow hydrology in Hydrus-1D:
The code assumes that when the air temperature is below zero all precipitation is in the form of snow, when the air temperature is above zero all precipitation is in the form of liquid, and there is a linear transition in between these two limiting temperatures (-2,2).
The code further assume that when the air temperature is above zero, the existing snow layer (if it exists) melt proportionally to the air temperature, i.e., the SnowMF amount for each day and each degree C.
This should be release with the Hydrus-1D update, hopefully this fall. I have not done anything for Hydrus-2D so far.

2. Animation needs to have a constant color spectrum for all time levels. It can not have different coler spectrums for each level. I think that make a lot of sense.

3. IN the latest version of Hydrus-2D we allow users to specify their own contour levels. However they still need to be at the same interval (clearly you can define the same colors for multiple intervals to achieve what you want). Here is a description on how that can be done:
This option is implemented under the command: Options->Scale Values. There the user can decide whether he wants to use the default scale or to define his own one.
The user scale can be read in from the text file, where on particular rows are numbers from the minimum to maximum in free format (see example below). The step between numbers must be constant, i.e.,: Value(N) = Value(0) + N*rStep.

Example of the text file:
-60
-50
-40
-30
-20
-10
0
10
20
30
40
50
60

## ID = 218, Flowing particles

Hi,
is there any option/trick to simulate particle transport with Hydrus- 2D?
Thanks,

Christiane

Christiane,
That would depend what exactly you mean by particle transport.
I have been working recently on an option that allows to simulate solute transport using attachement and detachment rates, with attachement rate possibly defined using filtration theory. We used this option to simulate transport of viruses:

52. Schijven, J., and J. Šimùnek, Kinetic modeling of virus transport at field scale, J. of Contam. Hydrology, 55(1 2), 113 135, 2002.

and microspheres (colloids):

68. Bradford, S. A., J. Šimùnek, M. Bettehar, M. Th. van Genuchten, and S. R. Yates, Modeling colloid attachment, straining, and exclusion in saturated porous media, Environ. Sci. & Technology, 37(10), 2242-2250, 2003.

77. Zhang, P., J. Šimùnek, and R. S. Bowman, Nonideal transport of solute and colloidal tracers through reactive zeolite/iron pellets. Water Resour. Res., 40, doi:10.1029/2003WR002445, 2004.

The other meaning can be particle tracking, which is a numerical technique for solving solute transport. We do not have that option.

Jirka

That sounds interesting
I have no instant access to the publications mentioned, therefore I would like to ask what does mean "Defining attachment by filtration rates". Does this option mean that material transport implies a change of the hydraulic conductivity to simulate colmation an decolmation processes?

Th deposition (attachment) coefficient [T-1], ka, can be evaluated from filtration theory as

Ks=3*(1-n)*e*a*v/2/dc

dc - diameter of the sand grains [L]
e - sticking efficiency (ratio of the rate particles stick to a collector to the rate they strike the collector) [-]
a - single-collector efficiency [-]
n - porosity [-]

We have, with Scott Bradford, also calculated reduction of conductivity and pore space due to depositing particles. But I do not think that we published that in any of our papers yet.

J.


Jirka,
Thanks for the hints. We sometimes have to deal with mechanisms of clogging at the interface of distinct substrates/filter materials and would appreciate having a more rational inspection of the complex interactions between hydraulic gradients, deposition / transport and the subsequent adaptation of conductivity and porosity. So I will track your publications on this interesting HYDRUS option.

Regards
Hector


## ID = 219, Gaseous diffusion

Hello HYDRUS users,
I have never used gaseous diffusion option of hydrus2d before.
When gaseous diffusion is modeled, where are gaseous concentrations stored in the output? Which file / which column? I could not find much information on this on the manual or the discussion forum.
In the graphical display of results, is the total concentration in gas and water phase displayed, or is just aqueous concentration displayed? I know that with sorption, just aqueous concentration is displayed, is that the case when exchange into the gas phase is also considered?

Thank you,
Defne


Defne,
Yes, it is the same as with sorption. Hydrus displayes only the liquid concentration. It does not print equilibrium sorbed or gas concentrations, since these can be calculated simply using the sorption isotherm or Henry's law. Hydrus prints only nonequlibrium concentrations, when considered, for kinetically sorbed solutes. Equilibrium concentrations (both sorbed and gas) are obviously considered in mass balance calculations.

Jirka


## ID = 220, Water flow parameters

Hello,
I am a new Hydrus user attempting to model some column test leaching for a class project. I am having difficulty finding Water Flow Parameters for the material I am looking at. The material is steel slag. I am assuming I could use parameters for a similar material such as sand and gravel but the examples in the program only go up to sand for grain size. The grain size for my material has a D50 of about 12 mm so the

particles are definitely larger than sand. Any help with some estimated parameters would be appreciated.

Sandy

Sandy,
Larger grain size materials will have tendency to have:
smaller residual water content (probably zero)
smaller porosity (you should calculate this from bulk density)
smaller alpha
larger n (4-6)
larger hydraulic conductivity
J.

The porous medium you are using is indeed very, very coarse (a median grain size of more than 1 cm!), so it must be a gravel. Do you also have fines between the gravel? If so, perhaps the approach by Raz Khaleel could be used (Water Resources Research, 33,8, 1875-1878, 1997). You then estimate the hydraulic properties of the fines, and adjust for gravel content as he showed.

I wondered if you were doing a variably-saturated flow experiment, or were leaching at a constant imposed boundary flux.

If variably-saturated flow, then your condutivity must be very high (assuming ponded surface), and you need to estimate the hydraulic properties as best as you can (e.g., very high Ks, and very alpha of about 0.1 1/cm of higher. The value n probably does then not matter as much (2 or higher as Jirka suggested).

If at constant flow, the main thing to estimate then is the water content (measure it before or after the experiment, or calculate from bulk and particle densities). The condutvitity is then not needed. Hope this helps.--Rien van G.

Hi
I have modeled crushed glass that has a D50 of about 12mm. I did a "soil water retention curve" and estimated residual water contnet, alpha and n. Let me know if you think that crushed glass is close to what you are looking for and if you are interested in those numbers.
Mazen

hello Sandy,
I also did a lot of measurements on Slags (MSWI), concrete aggregates and masonry agrregates( all secondary building materials). If the material you are meantioning is not seaved I foud that the smaller parts were controling the flow. Furthermore there is a problem of a substantional intraparticle flow, this requires a bimodal waterretention curve which is not possible in Hydrus.

the experiments I did were done due to the problem of the missing data
I could not find any data.
It is always best to measure data (difficult) because this has physical meaning
I can send you my data if you think this helps. (pressure plate experiments)
greets IJsbrand


Hi all,
Does anyone have water flow parameters for boulders or cobbles? Do you know any
paper that talks about that?
Thanks,
Maz.


hi all
I would also be very interested in MEASURED water retention data of boulders or
cobbles

Thanks IJsbrand


### ID = 221, Solute transport

Dear Jirka,
I want to compare nitrate leaching under a furrow system. One of the scenarios would
be apply the fertiliser as a solid (220 kg N/ha) and the irrigate under a furrow
irrigation. I want to use ponding depth as the boundary condition. How do I get the
conc. to enter in the solute boundary condition?

Thanks
Thabo


Thabo,
Look at the Furrow example in the Direct1 workspace. It is a nitrate flow from a
furrow to the subsurface drain. That should give you a very good idea on how to
model such system. This example is also described in one of our tutorials where you
can get step by step insturctions on how to build such project.

Jirka


### ID = 222, Inverse solution, anisotropy

Hi, I am user of Hydrus 1D, 2D, RETC, etc. I was in the last shortcourse from hydrus
in Hannover. I work with water movement in Hillslope and we consider the
anisotropy of the water conductivity. I have some cuestions for you and I hope that
you can help me.

1.- It is posible to have more than 10 observations points to work with the inverse solution. We had 18 tensiometers at 3 sites in the hillslope, and I think that it would be better if I can all of this information.

2.- Will you consider in your model that the anisotropie of the water conductivity depends on the matric potential, like suggested by Mualem (1984).

3.- We habe take undisturbed soil samples from the different soil horizons at the 3 sites of the catena. The samples were taken at an angle of 0, 45 and 90° to measured water conductivity and to determinate the water flow. It is posible makes changes on your consideration of the effect of the anisotripy on the water flow.

Jose


José,
a) In the latest version (that can be downloaded from this site) we allow up to 30 observation nodes.

b) I do not have any immediate plans to do that. I remember doing this in the past (in some applications with Gerret deRoij), but must have discarded the code (since I can not find it anymore).

c) Hydrus allows anisotropy to be at different angles. Each element can have different angle of anisotropy.

Jirka


**ID = 223, Nonequilibrium sorption**

Questions are related to output-files when calculating solute transport in a 1D soil profile with nonequilibrium sorption.

What are the dimensions and values in balance.out (Documentation and output files are sometimes contradictory)?
ConcVol - amount (M) or concentration(M/L3); is it the total amount of solute in the soil water?
cMean - ?
ConcVolIm (M/L2)- sorbed concentration? What is referred area (L2)?

And what is the reference value for Sorb(1,...,NS) or Csorb(1,...,NS) in NOD_INF.out (dimensionles values? or concentrations?)

Thank You!


I do not think that documentation and output files are contradictory. Units are as follows:

ConcVol [VM/L3]
ConcVlIm[VM/L3]
SorbVlIm[ V]
cMean [M/L3]
cMeanIm [M/L3]
sMeanIm [-]
Where V=L3 for 3D application, V=L2 for 2D application, and V=L for 1D applications.

From this it follows that ConcVol, ConcVlIm have units [M] for 3D, [M/L] for 2D application, and [M/L2] for 1D applications, and cMean is in [M/L3]. ConcVol is the total mass in the transport domain in equilibrium phase, while cMean is a mean concentration in the transport domain.
Since s (sorbed concentration) has units of [M/M]=[-], then integral of s over transport domain, i.e., SorbVlIm, must have unites of [ V], i.e., [L3] for 3D, [L2] for 2D, and [L] for 1D applications. SorbVlIm is the total mass in the transport domain in nonequilibrium phase (immobile water or kinetically sorbed chemical), while sMean is a mean concentration in the transport domain in nonequilibrium phase.

Transport domain is the length in 1D, area in 2D, and volume in 3D applications.
J.

## ID = 224, Solute transport

Hi,
I'm currently using Hydrus-2D to simulate subsurface drainage functionning with the presence of a saline groundwater. With the default parameters and space and time weighting schemes, the duration of simulations pass from few minutes (with water only) to more than two hours (water+solute).

Is there a possibility to reduce this duration playing on the weighting schemes or by an other mean?
Thank you in advance,
Sami
Sami,
If the simulation last only few minutes when you run water flow only, and several hours with solute transport, that most probably means that the time step is reduced due to the Courant number stability criterion (see time-information -> courant numbers). If that's the case, than there is only on option to make it faster, and that is to use courser spatial discretization.

If Courant numbers are smaller than 1, then probably stability (peclet*Courant) index reduces the time step. In that case you can either increase the stability index, ot increase dispersivity.

Jirka

**ID = 225, CXTFIT velocity?**

Hello Folks,
I'm trying to learn and use the public domain version of STANMOD. I'm specifically looking at using the interface with CXTFIT to calculate and model some transport parameters for some column data. Unfortunately, even with using the STANDMOD and the CXTFIT manuals, I'm having problems understanding some of the terms and inputting the correct units. I'm hoping that someone on this forum would be willing to offer some help - I would be highly grateful for any assistance!

I have tritiated water breakthrough values for some columns that have 30cm X 2.5cm dimension. The flow was established with a syringe pump set to 6 ml/hr. If I did the math conversion correctly the flow velocity for the column would be 29.33 cm/day. However, when I set this value and run the model and allow the program to fit the velocity it brings it down to 0.717? Also, if I do not allow the program to fit/adjust the velocity it is unable to fit the data.

I'm also having some confusion with the term "production" as used under the "type of production distribution" question. What is this? For my columns I've been using zero but I don't know if this is correct or not. And, under the section "parameters for inverse problem" I don't understand the fill in box for "position of the breakthrough curve." From some of the example column data I noticed they used a value of 1. However, since it is a 30cm column, shouldn't it be 30? But if I set it to 30, the modeling run gets unhappy and doesn't fit any values. Can anyone tell me what this value is for and the proper way to use it?

Thanks - DougW
You need to specify time and positions in the input either dimensional or dimensionless in Input and Output Data Code. Dimensionless parameters are defined in Table 2.1 and Table 3.1. In the case of L = column length (30cm), the dimensionless position at the bottom is 1. If you can not solve the problem of v after correcting Input and Output Data Code, please let me know.


> I'm also having some confusion with the term "production" as used under
> the "type of production distribution" question. What is this?

The production terms are described in page 11-12 & Table 6.9 in the
manual. If any further questions, please do not hesitate to ask me.

Regards,
Nobuo


**ID = 226, Element information error**

Jirka,
In a furrow irrigation system I am using a particular solute conc. as initial condition. I am running the model in two stages.

Stage 1. Furrow irrigation with time variable boundary heads and some solute in a patch
Stage 2. Import h.out and conc.out from the stage 1 and use as initial conditions for stage 2 but the boundary will be flux boundary condition (no flux).

I could run the stage 1 without any problems but when I start stage 2, an error message comes" Error when reading from an inputfile Domain.in Element information" could you help me to sort that out please?
Thanks
Thabo

Thabo,
You must have somehow corrupted the domain.in file.
Questions to ask?
a) Did you do the import without any reported problems?
b) Did you do the import before modifying solute transport parameters, such as changing it from equilibrium to nonequilibriu,?
c) Can you still open this file with the graphical interface, i.e., can you open the Boundary module and look at initial conditions?

If you can do all this then I'm at a loss, and would need to see the input files.

Jirka

**ID = 227, Final output graph**

I am a new user of Hydrus 2D software.
I am using the column experiment as a model. Some of the assumptions made are steady state flow, saturated column, no adsorption and degradation. The dimensions of the column are 54cm height and 10cm length. The pressure at the bottom of the column is 60cm and at the top is 0cm. water along with a solute is made to flow through the column. The fixed initial solute concentration of 10mmol/cm3 is maintained at the inlet.

I am able to get the graph for outlet concentartion vs length. but am not able to get the graph between concentration and time.

How can i get this? I saw few example problems producing default output graphs for concentration vs time. But i didn't get anything like that.

thanks
mani

Mani,
You need to specify an observation node at the bottom of the transport domain. Then you will get a breakthrough curve with the "Observation Points" graph.
Jirka

**ID = 228, Peclet and courant number**

I have a couple of questions. I do know the answer but am not very confident about it. so i want to clarify it.
How does the courant and peclet number works in Hydrus 2D? How can I alter these numbers?
Is it by spacial discretization? OR
Is there anything to do with the advective velocity (Vx), average grain diameter (d), and coefficient of molecular diffusion (Dd) parameters.
mani

mani,
Look in the manual for definition of the Peclet and Courant numbers. From these definitions it follows that:
a) Peclet number may be decreased by
- using finer spatial discretization
- increasing dispersion
- decreasing fluxes
b) while Courant numbers can be decreased by:
- decreasing time steps
- decreasing fluxes
- increasing sorption
- increasing spatial discretization
Peclet number relates mainly to spatial discretization, while Courant number to temporal discretization.

Jirka

**ID = 229, Solute transport**

I would like to submit a query relating to Hydrus 2d.
field experiments were carried out on two type of soil Sand and sandy loam to investigate the effect of emitter " surface point source" discharge rate, volume of water applied on wetting pattern. Solute distribution was also investigated where various concentrations of Na Cl were added within irrigated water. for both soils, the results were well compared for moisture distribution but regarding to the solute distribution, the simulated concentrations found to be higher than actual concentrations.

I wonder why and how?
Is it related to solute transport parameters? The Longitudinal dispersivity "3 cm" was determined in laboratory using saturated column. The transverse disversivity was assumed to be 1/10 of longitudinal dispersivity "0.3".
The initial moisture content of soil was very dry "around 0.02".
The initial solute concentration in soil was very low "assumed zero".

However, when I chose 10 cm for Longitudinal dispersivity and 1 cm for transverse disversivity as mentioned in the example accomplished to Hydrus 2d model "subsurface source" the results showed acceptable comparison.

Is it true to assume the same value of longitudinal dispersivity and transverse disversivity as mentioned in that example since longitudinal dispersivity was used for big system size.

Yours Khmaj


When measured concentrations are smaller than simulated concentration calculated using measured parameters, then the overestimation is given either by
a) inadequacy of the measured parameters,
b) inadequacy of the conceptual models, i.e., some parameters are unaccounted for.

It is very common that field values of longitudinal dispersivity are larger than measured values in the laboratory.

Jirka


## ID = 230, Inverse

I need some help with inverse modeling. I have infiltration rate data for a falling head infiltration test and am trying to determine the hydraulic properties of a soil. I cannot just assume saturated conditions because there is a 2cm low K layer overlying a high K layer. What are the x, y, type, and position codes I need to use in the Data for inverse solution screen?

Thanks,
John



John
x - time
y - infiltration rate or cumulative infiltration
Type - 3 for infiltration rate and 0 for cumulative infiltration
Position - 1
Weight - 1
Next time before you pose questions in this discussion forum, look at least in the online help where all this is explained.

Jirka


## ID = 231, FAQ No 25

Hi Jirka,

About l in the soil Hydraulic properties:

This is the tortuosity factor in the hydraulic conductivity function. It was found to be 0.5 by Mualem, but there is not a strong agreement on this. People in our lab (Marcel Schaap) claim, after analyzing huge UNSODA database, that it should be more like -1.

Can you please explian how it can be negative?
Do you expect to use low value such as 0.2 for clay and like 1 for gravel?
Thanks,
Mazen


Mazen,
The l parameter in the Mualem-van Genuchten formulation of the hydraulic conductivity function in theory represents the tortuosity factor, and thus should obviously be positive. In majority of applications, it is, however, simply used as a fitting factor, and then its actual value has no physical meaning. That's why it can be negative.

One should, however, be careful, how negative values to use. Very negative values actually lead to the increasing conductivity with decreasing water contents, clearly physically impossible.

Jirka


## ID = 232, Mass units

dear users
I am not sure how to choose the unit for the mass, and how related quantities are thereby affected. For example, the soil bulk density (first solute transport parameter) could be 1.8 Kg/cm^3 or 1800 g/cm^3, or 1.8*10^9 g/m^3, as it depends on the units chosen for mass and lenght. However, I saw examples in which no unit was chosen for the mass, and I also verified that the code runs even if you type a fake name in the box for the mass units, like "frank". What would the unit for the bulk density be in this case? "frank/m^3" ??

thanks for your attention
Paolo Castiglione


Paolo,
That's correct. If you choose mass units as "franks" than the bulk density units will be "franks/m3" (and the same for concentrations). (I hope you are not trying to francofonize the Hydrus code though .)

But you must remember one thing. Hydrus needs consistant units. Since mass units appear in several terms of the CDE equation, all these terms must be consistant. Then,

for example, the Kd coefficient must be in "m3/frank" (gamma_w - frank/m3/s). Once you keep units consistant, you can use whatever you want.

Jirka

## ID = 233, Falling head axisymmetrical 2d flow

Hi,
I would like to know whether HYDRUS 2D can model axisymmetrical 2D flow from a disk inserted in the soil under falling head conditions.

The problem deals with the diffusion from a tube (inserted in the soil) of radius r1 open at the bottom and filled of water up to a high h at t=0. Can HYDRUS 2D model water infiltration into the soil under falling head conditions?

Carlos

Carlos,
Hydrus can simulate infiltration at changing head conditions (including axisymetrical 3D problem). However, the heads versus time need to be specified at the input. I understand that you probably need a BC that would automatically adjust water level in the tube (and thus the pressure at the boundary) to the infiltrated amount. Such system dependent boundary condition is not implemented in Hydrus, although it would not be too difficult to do that.

Jirka

## ID = 234, Precipitation and evaporation

Hello!
I am a new Hydrus-2D user and I have some questions to the forum. I am trying to simulate water movement in a road construction. I have defined the roadsurface as impermeable (zero-flux) and I only allow the precipitation to infiltrate in the road shoulders. The roadsurface leans towards one side of the road so that the total surface runoff infiltrates there. Accordingly, I have to define different boundary conditions for the road sides.

I am using atmospheric BC on one roadside with both precipitation and evaporation and on the other roadside I am using variable flux (evaporation-precipitation). I am using the version which treats variable flux as atmospheric BC.

Now, I have two problems:
Firstly, I am having problem when the evaporation has been larger than the precipitation (actual flux > 0) and it switch back to a situation when the actual flux <0. HYDRUS tells me that there is a floating point error.

Secondly, the figure of variable flux in the output: Water Boundary Fluxes is showing higher absolute values of both evaporation and precipitation, compared to the values that I have defined in the input file.

I use a space discretization of about 2 cm at the surface and have used many variations of the time steps.

Karin

Karin,
Problems:

Second problem: Obviously the reported fluxes for the variable flux can not be larger then the sum of evaporation and precipitation. However, did you check the units? While for the atmospheric BC Hydrus report the flux in L/T units (i.e., average flux across the atmospheric BC), for the variable flux BC is reports the flux in L2/T units, i.e, the total flux across the BC (flux*length). Isn't this the reason?

First problem: When the soil surface is large and there is a sudden precipitation, the code needs to use small time step. To allow small time steps, I usually specify the initial time step equal to 0.0001 d, and the minimum allowed time step around 1-2 s.

By the way, I'm working on an option to simulate overland flux together with Hydrus-2D. This is a work in progress, and unfortunatelly at present not too stable.

Jirka

**ID = 235, Unity in export all function**

Dear Hydrus user's
I'm a biginner using hydrus 2 d software. I woulde like to exploit results fron velocity and pressure Hed value from the commanfd export all from graphics.
I do not understand the value returned: There is 4 columns of valmues and five head of column name (index X, W1, W2, W3, W4, P).
I would like to know wich value column refered to which head, and what this variables mean.
Moreover, I would like to know how can I set the color scale to be full in my results scale of velocity and pressure head.

Best regards
Keats

Hi Keats
column heads (W1,W2,...) in Hydrus versions prior to 2.102 were really confusing and there was no relationship to names of Hydrus quantities. It has been fixed in the

latest version 2.102 and now there are already correct names. Columns actually contain (and always contained):
1. index
2. x-coordinate
3. values of quantities that you can find in the combo-box for selection of current quantity in the module Graphics (in the same order).

I don't understand to the second part of your question but if you want to create your own scale please use command Options->Scale Values. Then you can load your own scale values from a file - see topic 217 - "Snow" (http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=217) for more details. But if you want to use this option you will have to download the latest version because this function doesn't work correctly in previous versions.

Regards Mirek


## ID = 236, Porosity

Hello,
I am a student using Hydrus 2D for a class project. I'm training on an example and I have to fix the porosity of the soil. I can't find where I can set the value in the software. Can you help me?
Thanks,
Rackham.


Porosity is equal to the saturated water content in the soil hydraulic properties dialog window.
J.


## ID = 237, Plot of outlet conc vs length - column experiment

I have modeled a column. The column is packed with sand and then saturated with water. Water is made to flow through the tube at a steady rate. A solution containing a tracer is then introduced into the sand column in place of the water. The initial concentration of the solute in the column is zero, and the concentration of the tracer solution is Co. The tracer in the water exiting the tube is analysed.

The Hydrus 2D and the analytical results for Outlet concentration vs Length were compared for different time periods(ex: 1, 2,3, 5 and 10 days)

When superimposing the graphs of Hydrus 2D and the analytical results for different time periods

am getting the following results
1 day - Identical
2 days - 10% deviation
3 days - 25% deviation

5 and 10 days - a bit more deviation
So I need some guidance and help to make it identical for all time periods.
What do I have to do inorder to get it right?
mani

1) Make sure that you are using the same type of concentrations. Hydrus uses the resident concentrations, while many analytical solutions are derived for the flux concentrations. Some analytical solutions hold for flux concentrations for one type of boundary conditions, and for resident concentrations for another type of boundary condtions.
2) Ceck the mass balance error. If it increases, try to use finer spatial and temporal discretization.

Jirka

## ID = 238, Solute concentartion drops down to ZERO

I have modeled a column. The column is packed with sand and then saturated with water. Water is made to flow through the tube at a steady rate. A solution containing a tracer is then introduced into the sand column in place of the water. The initial concentration of the solute in the column is zero, and the concentration of the tracer solution is Co. The tracer in the water exiting the tube is analysed.

These are the boundary conditions for the column experiment

Flow equation boundary
- Bottom of the column ( Pressure 65 cm)
- Top of the column (pressure 5 cm)

Solute equation boundary
- bottom ( fixed concentration of 10 mmol/cm3)
- top ( variable head boundary)

The final output graph is between the outlet concentration and Length of the column.

In this plot the concentartion of the solute drops down to ZERO at the end of the column length irrespective of the time period. but this is not the case in reality. There should be some amount of solute concentration within the column even after the completion of the run time.

Is there any reason for this happening? How can i rectify this?
mani

## ID = 239, Large precipitation

Dear all

My question will seem familiar to many of you but I can't find the right solution to my problem.

I have a large change in the precipitation rate between two consecutive days (i.e. from 0.44 to 3.24 cm/day). This steep increment is causing instabilities and run time errors causing the program to stop.

I tried to manipulate all the factors that can attenuate this effect, and I'm still not able to overcome the problem.

I reached the following:
Initial time step = 1E-12 day
Minimum time step = 1E-30 day
Theta tolerance = 0.001
Head tolerance = 1cm
Minimum mesh size = 0.6cm (between layers)
Mesh size at surface = 0.9cm
Maximum mesh size = 2.5cm (mid layer)

Sofia

Sohpia,
I would not use such small time steps. I would recomend:
Initial time step = 1E-4 day
Minimum time step = 1E-7 day
The other parameters seem fine.

What is the soil. If you have clay with n<1.2, use option with air-entry value of -2 cm.
Jirka

dear sofia,
This topic was listed before. But that does't matter. I had the same problem a midsummer thunderstorm gives the same results. You can do something that is no the truth but can overcome your problem

You can smoothen the rainfall over a few days but keep the amount of precipitation the same. (so instead of 0 0 0 10 0 0 you get 0 2 4 3 1 0) So if it doesn't matter in your simulation (if you take long interfalls) you can do this.
greets IJsbrand
Thank you for replying
Re. the soil type: I have two layers. The surface soil has an n=2 and the bottom soil has an n=1.05 and I'm already using the -2cm air entry option.
Re. smoothing the precipitation rate over the time interval. I tried this previously and it worked fine. The problem now is that I need exact values because I'm trying to find the effect of daily storms on a landfill cover systems. I'm intereseted in the changes in precipitation rates rather than the overall total volume of rainfall.

Sofia

Sophia,

Your lower layer has n=1.05. That looks like a very compacted clay that will also have a very low conductivity, that probably results in water layer building up on top of it. You can increase the pressure head tolerance to 1-2 cm (this has no effect on mass balance error, since it is used only in the fully saturated domains where water content is constant and equal to porosity). That may stabilize the solutions, since one does not need to care about small fluxtuations in the positive pressure heads..
Jirka

Dear Jirka
I increased the head tolerance up to 3cm and the application was successfully run. But I noticed at some time levels a mass balance error of 2 to 3 %.
Is it an acceptable range?

Thanks for your valuable support
Sofia

Sophia,
I can't say. That depends on your application. I prefer to have mass balance errors below 1%, but I could probably live with 2-3 also.
Jirka

**ID = 240, Solute transport**

Hello everyone! I hope somebody will be able to help me. I am modelling a pilot-scale field experiment of soil remediation. I have a variable groundwater level and groundwater moves with velocity 0.5 m/year. But I can't put these two factors in one model. For initial flow conditions I assigned "linear distribution with depth" -220 to 680 and "slope (tangentum alpha)" -0.0001. When I set left, right and bottom boundaries as "constant pressure head", so this is no variable groundwater level, the plume moves as I expected 50 cm from left to right, but when I set left, right and bottom boundaries as "variable pressure head" with the same initial conditions, the plume stays right in the middle, the groundwater level is changing thoughout the simulation but these is no gw movement. What problem could it be?

Thanks

Marina,
If you specify on both sides of the domain the same time variable head BC, then heads on the left and right side of the region are the same and there is no flow. Therefore the plume sits in the middle of the region and is not moving.
Jirka

Jirka, thank you very much for your reply. But I still don't understand how to solve this problem. When I specify on both sides of the domain the constant pressure head, I assign the certain number for each node. As a result I have GW flow, but GW level

stays the same. But for variable pressure head I am not assigning any numbers in Boundary Conditions Editor , only in F(t) Variable Boundary Conditions in the column GWL. I thought that if I specified in initial conditions "slope" it would work, but it doesn't work. Is it possible to make variable GWL and GW flow work in one simulation?

Thank you.
Marina

Marina,
Hydrus can handle only one time-dependent head boundary condition, and this value (GWL) is assigned on the entire specified BC (taking into account hydrostatic distribution on the boundary => therefore there is no flow if you apply it on both sides). If you send me an email, I can send you an updated version, in which you can specify on one side time-dependent head boundary condition (and thus force the GWL to fluctuate) and on the other the head gradient (paralel with the slope).

Jirka

Jirka, thank you very much. I would really appreciate if you could send me the update. At this point I thought that I would run two simulations without groundwater flow: one with variable groundwater level, the other with constant groundwater level. Hence I would show the effect of groundwater fluctuations on the plume. Then I would run the simulation with groundwater flow and for the final results take into account the difference between first two simulations.

My e-mail: laferom@mcmaster.ca
Thanks
Marina,

I have emailed you a version of the Hydrus-2D programthat will allow you to specify variable head BC on one side of the domain and gradient type BC on the other. Therefore specify the variable flux at one side (associated with GWL in time-variable BC), zero flux at the bottom, and free drainage on the other side. The code will ask you to enter the gradient for this side, once you start it. Let me know how it worked. Gradient is tangentum.
Jirka

**ID = 241, Solute transport**

Hi,
I want to simulate the solute transport in a lysimeter. By comparing the model outputs pressure head and water content in several depths and bottom flux versus time for two model runs- one just with waterflow, the other with considering both waterflow and solute flux- I discovered differences. So it seems that considering the solute flux has some influence on the waterflow. I didn?t found something about this in the manual.

Does Hydrus account for the osmotic potential or the influence of the solute on the viscosity?

Unfortunately I got some soil frost during the time of the tracer experiment. Is there any possibility to consider the freezing and thawing of soilwater in HYDRUS1D?

Although the lysimeter is filled with sandy loam the measured BTC?s give evidence that there is preferential flow and in the version 2.02 of Hydrus I found some soil property models that might be useful in this case. Unfortunately they are switched off. Because I know that in the earlier version of HYDRUS the soil property models ?Dual porosity?, ?Kosugie? are working I thought it might be that there is something wrong with the installation (some old models with dual porosity are running with Version 2.02)?

Thanks for your time
gusti

Gusti,
a) There is no direct effect of the solute transport on water flow in Hydrus. There is only an indirect effect via root water uptake, if you consider an option that uptake can be decreased due to high concentrations.

b) Freezing/thawing: We have recently been working on freezing/thawing module for Hydrus with Klas Hansson. This will be published in the next issue of Vadose Zone Journal (76. Hansson, K., J. Šimùnek, M. Mizoguchi, L.-Ch. Lundin, Water flow and heat transport in frozen soil: Numerical solution and freeze/thaw applications, Vadose Zone Journal. (submitted August, 2003, accepted).). This option is not yet available to general public, but will be in the future versions of Hydrus.

c) Preferential flow: We are currently working on a suite of different approaches to preferential flow (see Šimùnek, J., N. J. Jarvis, M. Th. van Genuchten, and A. Gärdenäs, Nonequilibrium and preferential flow and transport in the vadose zone: review and case study, Journal of Hydrology, 272, 14-35, 2003.). Again this option is not yet available to general public, but will be in the future versions of Hydrus. Other models of soil hydraulic properties (other than van Genuchten-Mualem and Brooks and Corey) will also be available in future versions. In the current version you can use the mobile-immobile water concept to simulate nonequilibrium/preferential flow.

Jirka

Hi,
So do Gusti, I want to simulate the solute transport in a lysimeter and I observed the solute flux has some influence on the waterflow, when the mobile-immobile option was used. I tried to change the maximum allowed concentration for Root Water Uptake (cRoot)from 0 to high values, but it did not change the result. Is there a threshold value for thetImob depending on total water content?

Thank a lot for your time
Marine.

Marine,
As I wrote in my first answer, the only influence of solute transport on water flow is due to reduction of the root water uptake due to high osmotic pressure (if considered). An indirect influence is through the time stepping routine, i.e., Hydrus can select different time steps when solute transport is considered. This, however, should not lead to different results, if the iteration criteria and time steps are selected correctly.

Immobile water content: This is constant during the simulation, and thus you need to make sure that immobile water content is smaller than the minimum total water content during the entire simulation. If this is not the case, then one can not define the pore velocity, since mobile water would drop to zero, or even negative values.

Jirka

## ID = 242, Assign material properties along diagonal

Hello,
Is there an easy way to assign material properties along a section bounded by a diagonal line? For example, I would like to assign separate hydraulic properties to soils above and below the diagonal line. I am currently using numerous rectangular sections in the boundary module to assign properties to each side of a diagonal interface. This is time consuming so I am hoping to find a more efficient way to accomplish this task. I searched the FAQ and previous postings but did not find a solution to this issue. Any suggestions would be appreciated.
Eric

Hi Eric,
try command "Edit -> Select by Rhomboid"
Mirek

## ID = 243, 2D model - floating point error
I have modelled a 2D aquifer with 200cm length and 10cm height. The aquifer is fully packed with sand (for example). I am draining (assuming it as well) the water from the middle of the aquifer. All the three sides of the aquifer has no flux boundary and atmospheric boundary on the top.

I am getting 'floating point error'.
what do i have to do to correct this error?
valliyappan.t

## ID = 244, Arsenic speciation

I am looking at the mechanism (sorption and degradation) of arsenic release to ground water. I am doing column experiments and have modelled the same using Hydrus 2D.

Keeping this as the base am planning to proceed with real systems (aquifers) using Hydrus 2D.

I would like to know whether it is possible for me to use Hydrus 2D to look at the arsenic speciation in aquifers?

If so, can you give me some suggestions please?
mani

mani,
No, Hydrus can not handle speciation reactions. Unless you can somehow reformulate the problem so that you can describe it using first- or zero-order terms, then it can not be modeled using Hydrus.

Jirka

## ID = 245, 2D model - aquifer design

I have modelled a 2d aquifer using Hydrus 2D. The dimensions of my aquifer are 200 cm length, 10cm height with a small disk(acts as a seepage) in the middle.

when I ran the stimulation, I got a floating point error.
Then I changed the parameters one by one for each run. But couldn't make it right.
Then I took the Furrow example problem from Direct folder. I changed all the input parameters with my data. But I didn't change the aquifer design. But I did change the boundary conditions.
I got the results for my problem.
Is there any catch in designing an 2d aquifer using Hydrus 2D?

Thanks for your time and help.
Mani

## ID = 246, Chloride transport

I am working on Aquifer Storage Recovery wells. The Underground aquifer at 54 m is brackish having 261 m.e/l chloride and we are recharging with a water of 1.4 m.e/l. I am taking variable flux at the cavity and a contant pressure equilibrium from lowest point on the right side boundary condition. In solute transport BC I am taking third type on cavity as well as on right hand boundary and at cavity the pointer of vector BC is 1 and at right side its 2. then in time variable condition I am giving cvalue 1 as 1.5 and cvalue 2 as 261. There are six soil layers upto that cavity and cavity is leaky. With these conditions the programme is running but not giving any output. can you expalin whats the problem or how should i give BC conditions and the conc.

With regards

Yashpal

**ID = 247, Topic unknown**

This isn't a discussion forum

**ID = 248, Run time error**

Hi,
I created a flow model, similar to the model in the tutorial - session 1 A. While running the model, I receive the following message in the DOS window:
" floating point overflow "

Does anyone have any idea?
Thanks,
Hila

**ID = 249, "Export all" error ?**

Hi,
We may have found a small bug in Hydrus 2.101 and wanted to alert you. In the Graphical Display of Results window, we have been using the "values along cross section" and "export all" options to produce ascii data for calculations and additional analysis. It appears the "export all" option exports the selected quantity (e.g. pressure head or concentration) for the selected print time, and exports all other quantities for the final print time for the simulation.

Regards Chris

Chris,
I have been thinking about how you can get around this.
This reason for the "Export all" option exporting variables at different time levels is that different time levels are associated with each variable, depending which time level was displayed last for a particular variable (default is the last time level). Thus if you want to export all variables for the same time level, you need to display them all first for this particular time level (just switch from one variable to the other at the same time level, before doing the cross-section), and then do the export.

Jirka

Hi
This problem has been fixed in version 2.102 (May 2004)
Mirek

**ID = 250, Pesticide simulation**

Dear All
In the case of pesticide modelling, from a simple case (no chain) to more complex one (with chain), like Pang et al 2000, Hydrus allows to include degradation (half life) with 1 order kinetic in liquid phase and adsorption liquid to solid (linear q=k*c). How can we check and assess the mass balance of degraded and sorbed pesticides ?
Is the column "sorb" in solute.out only used for the case of mobile/immobile exchange ?
Is it possible to take into account the degradation of sorbed pesticide by the way of the μ's in solid phase or μ'w in liquid phase ?
And last question, can we incorporate desorption (with k<0, but simulation did not run) ?

Best Regards
Julien


Julien,
a) The column Sorb reports concentration in the nonequilibrium phase, which is either concentration in the immobile zone (when M-IM model is used) or concentration at kinetic sorption sites (when two-site sorption is used).

b) The mass balance of degraded pesticide is reported in the column cvCh1 in the solutex.out file (cumulative amount of first-rder degradation).

c) Is it possible to take into account the degradation of sorbed pesticide by the way of the μ's in solid phase or μ'w in liquid phase ? Yes.

d) Negative Kd would mean that there is exclussion (anion exclussion) of solute. Hydrus should handle negative Kd values, such that the retardation factor (R=1+ro*Kd/theta) is still positive. Negative retardation factor is not physically possible.

Jirka


**ID = 251, Water transport in columns with specific dimension**

I'm supposed to use Hydrus 1D for modelling water and solute transport through columns. The columns I'll be using are 100cm L x 5cm I.D. Is there any way to specify within Hydrus the dimensions of the column used?

Any help will be greatly appreciated.


Hydrus-1D is a one-dimensional program, and thus it deals with only one dimension. Consequently, you can specify the depth of the profile, but not dimensions in the other

directions (x and y). If you need to convert results of a simulation into larger scale, just multiply boundary fluxes with the surface area that the simulation represents.

Jirka

## ID = 252, Solute transport

Dear all,
I am trying to simulate KNO3 transport in a andisol (field, columns).
Water :
I measured the hydraulic parameters (Wind, Beer-Kan and Decagon) and the water flows seems to be correctly simulated.
Solute :
I am beginning this part and I am not sure to have the appropriated parameters (and my questions are basics...).

1- For solute Specific Parameters (Molecular diffusion coefficient in free water and Molecular diffusion coefficient in soil air), I had a look in the HandBook and they only specified diffusion coefficients of KNO3 (1.846) in aqueous solutions. Is that correct? Do I have to correct it with the tortuosity? Does volatilization take into account the diffusion in the air? Do not we have to specify an independant coefficient for each species (K and NO3?)?

2- For dispersivity, I tried to use LiBr in soil columns to have a representation of the "tracer" flow but there was an interaction between the exchangers (anionic and cationic) and LiBr. Is it necessary to use isotopic tracers (of H2O?) and the equation of "convection-dispersion" to determine this parameter? And is it necessary to have saturation?
Someone told be to try D=z.v with v the cinetic et z the dispersivity. z would be a lenght and could be characterized by the size of the "heterogenity" or aggregates. But it is not a "purist" way of doing, isn't it?

3- For Solute Reaction Parameters, it's OK. I hope I could introduce Langmuir-Freundlich parameters for an anionic and cationic sorption (batch experiment).

Thanks for your answer(s),
If you have got references about the second point, it would be wonderful!

Julie

Julie:
1.9 ($10^{-6}$ m2/s?) is a good value for the diffusion coefficient in water. Generally about 1 to 1.6 cm2/day). Yes, multiply with the tortuosity factor (about 0.6). That should be good enough for most applications. A good reference is the Flury and Gimmi paper in SSSA's new "Methods of Soil Analysis" monograph (chapter 6.2). Diffusion coefficients in air vary more, and are much higher. Are you working with a volatile chemical?

It is not necessary to use perfect tracers to get an estimate of the dispersivity. For many transient flow and/or field applications, use as a good initial estimate the approximate rule (as shown in studies by Anderson, Jury, Gelhar and others) that the longitudinal dispersivity (z in your notation) is one-tenth of the transport scale (for 2D use for the transverse dispersivity 0.1 of the longitudinal dispersivity). The HYDRUS codes automatically adjust the dispersion coefficient as D=z.v (your notation). You may want to use slightly higher values for z when the soil is aggregated or structured in some way or another, and somewhat smaller values if it is a very homogeneous soil. These estimates are generally good enough for most transient flow/transport problems. Results are more sensitive to the dispersivity when your flow regime is close to steady-state over long periods of time (i.e., constant water fluxes and water contents).

Yes, use Langmuir, Freundlich, or the combined Langmuir-Freimndlich isotherm, whatever fits your batch data best. Good luck.—Rien van G.

Dear Rien van G.,
Thanks for your answer!
1,9 is in 10^5 cm².sec-1 à 25°C in the handbook. It might be an error somewhere. I will have a look in other references.
Actually, I am not working directly with a volatile chemical but I thought that the first-order decay chains could be bidirectional and simulate denitrification for example (NO3>NH4>NH3 ?). That's why I was searching for a diffusion coefficient in the air.
Concerning the dispersion coefficient, I am wondering if I not getting lost! Well, I thought that – if d = dispersivity = z, Dm = dispersion coefficient (mecanic) = v.z, Df = diffusion coefficient = Do.t and D = dispersion coefficient – then: D = v.z + Do.t, I am wrong? I am going to put longitudinal dispersivity = 3 cm (and transv.disp = 0.3) and see what happened if I put a smaller or higher value.

Regards,
Julie

The Diffusion coeffcient for NO3 should be about 1.9 x 10^-9 m2/s (1.6 cm2/day) (See Flury and Gimmi Table, or some other source).

The decay chains in Hydrus go only one way.
Yes, add the two terms for D (D=vz+Dot).

Are you using HYDRUS-2D for the column experiments? Perhaps use the simpler HYDRUS-1D (no need to use transverse dispersion for a small column), unless perhaps you have an equivaluent 2D field setting to which you want to apply the same simulations. Good luck.--Rien.

**ID = 253, Output -> Input**

Hello everyone,

I simulated the solute transport, so as an output result I have a big plume sitting below the groundwater level. Now I want to simulate the "pump & treat" technology. How can I make the output files from the first simulation input files for the second?

Marina Mironov

Marina,

Make a copy of the project using the project manager, and then in the boundary module go to the command "Condition->Initial Condition->Import pressure. You get a "Open File" dialog. Find the h.out file that contain results of the original simulation, and select which time level you want to use as initial condition. Then do the same for the concentration. You can then modify boundary conditions, and change any other variables. Only the mesh must stay the same.

Jirka

## ID = 254, Freundlich isotherm (beta values)

I have modeled a column. The column is packed with sand. Water is made to flow through the tube at a steady rate. A solution containing a tracer is then introduced into the sand column in place of the water. The initial concentration of the solute in the column is zero, and the concentration of the tracer solution is Co. Assuming Freundlich adsorption isotherm. The tracer in the water exiting the tube is analysed.

The model was run for various Beta values ranging from 1.25 to 0.47. The BTC were obtained and it illustrates the following

Beta > 1.0 ----- More adsorption than linear
Beta = 1.0 ----- Linear adsorption
Beta < 1.0 ----- Less adsorption than linear (upto 0.7)
Beta < 1.0 ----- More adsorption than linear (below 0.6)

For Beta value less than 0.6 the adsorption should be less than 0.7 but it isn't. I checked the input parameters and it was correct.

Is there any reason for this to happen? Any suggestions please.
mani

Hello Mani,
if your concentrations are really low, there exists a range in a Freundlich isotherm where sorption for the nonlinear case is greater than for the linear case. You should plot your isotherms and look at the concentration range you use in your modelling.

Heinke

Mani, Heinke:
That is right, to some extent. When your Co is less than 1 (in whatever units you use in your calculations), sorption increases with smaller n (look at the average slope of the isotherms between 0 and Co). When n is about 0.6 or less, the transport problem is becoming very nonlinear, with the front of the BTC becoming very steep. You may want to check the effect of using smaller time and spatial discretizations on the results. Thanks.

The above results are the reverse when Co is much greater than 1(more sorption with higher n). When n is only slightly higher than 1, things depend also on whether you have continuous injection, or add a short pulse. Hope this helps.--Rien van G.

## ID = 255, Flux (time variable)

Hi Jirka,
I am simulating water injection through a pipe using a constant flux. The simulation ran fine. However, if I switched to use variable time boundary flux for the same simulation, the simulation shuts down. The same simulation works fine if I used constant head or variable time boundary head. I couldn't get it to work using variable time flux!!

Is there something I should be careful about (using water content or pressure head as initial condition?) or something has to do with hcritA? I was using hcritA = 10000000 for zero flux.

Mazen

Mazen,
I can think of one thing. While for the constant flux BC, the fluxes are positive for infiltration and negative for outflow, they are negative for infiltration for the time-variable flux BC (see the help file, or the drip example in the direct1 workspace).

Jirka
## ID = 256, Unit consistency

I am giving the units only in three dialog boxes of Hydrus 2D software; which are as follows
Geometry information dialog box - L (cm)
Time information dialog box - T (sec)
solue transport dialog box - M (Kg)
This is particularly for mass units, if i choose the solute concentration in kg, what will happen to the bulk density term?

mani

Mani,

Look at the governing equation. From that it should be clear how the units are handled. Since the bulk density is multiplied by the Kd parameter and some of the decay constants, you need to have consistent units for the parameters. Concentrations units also must be consistant, altghough they may be different from those used for the bulk density. The M (Kg) variable entered in the solute transport dialog box is only a string that has no consequence on the solution. But the consitance of units described above is important.
Jirka


Dear Jirka,
I looked at the example problems (5 and 6). In those problems, the units of some of the input parameters were

Bulk density - 1.222 g/cm3
Initial solute concentartion - 20 mmol/lit
Adsorption co-efficient - 1.14 cm3/g

so in this case two different mass units have been used, two different volume measurement units are used.
But in the program the co-efficients (1.222, 20, 1.14) were not changed. They all remain the same.
I can see the consistency between the bulk density and adsorption coefficient. But the initial conc doesn't have same units; which is also a part of the adsorption equation.
So what approach do I have to follow solving problems?

mani


Mani,
If you multiply the bulk density with the distribution coefficient, you get a dimensionless result. Then the concentration can have any units you like, since it appears in all terms of the CD equation.

Jirka


**ID = 257, Need help to program**

Hi anyone,

Can someone help me with a program that I am writing that will solve the CDE analytically for continuous inputs of nitrate in order to calculate the concentration of nitrate leaching below the root zone.

IF someone can help I would greatly appreciate it.

Jean Jolicoeur
jjolico1@binghamton.edu.
Binghamton, NY

**ID = 258, Time running over**

I have setup HYDRUS 1-D to run for 18 days and under certain conditions, it passes the 18 day time and keeps running. Has anyone seen this?

Please check if the units you r using are consistent. Also pls check your max number of iterations. Though I have not experienced this problem before, I am assuming that either you r specifying 1 set of units and using another.

For anyone who has seen this problem, it appears that if your min. dt is to small, the program does not stop running unless stopped manually. No
data seems to be lost though.

**ID = 259, Retention curve for gravel**

Hello!
What would be the best approach to analyzing the hydraulic parameters for course grained material?
I've been using the Rosetta prediction model for the calculations of the parameters for fine grained materials. This model only allows the input of clay/silt/sand percentages. What should I do when I have a material consisting of 30% of sand+silt+clay and 70% of gravel?
Has anybody done any studies regarding retention curves and hydraulic parameters for coarse grained materials?

Thanks,
Jure

Dear Jure:
The best seems to use Rosetta to determine the retention properties for the fines (or measure it), and then to correct linearly for stone or gravel content (thus assuming that stones and gravel do not retain water (or do not change their water content)). A good example is the paper by Khaleel and Relyea (Water Resources Research 33, 8, 1875-1878, 1997), who showed that corrections based on bulk density ratios worked somewhat better than volumetric corrections (at least in the wet range). Plus look are references cited in that paper. The correction for the hydraulic conductivity is more complicated, as indicated for example by Bouwer and Rice (Ground Water, 22, 696-705, 1984).

**ID = 260, Alpha value in soil hydraulic property**

Hello,
I wish someone could answer this question:
Could the alpha values used in HYDRUS be less than one and greater than 15 (in the units of per meter)?

I ask this because the alpha values listed in RETC manual are all greater than one but less than 15 per meter.

Thanks for your help.
Liping


Liping:
I am not sure my previous answer went through. Yes, alpha can be easily less than 1 (1/m) if the soil is very fine-textured (a sticky clay). Bet Netofa Clay had an alpha value of about 0.15 in our original 1980 paper. Rocks have even smaller values. Remember that alpha is roughly the inverse of the air entry value. Similarly, alpha has no upper bound. Alpha for very coarse soils (those tending towards gravel) can have alpha values greater than 15 or more, although I have not seen many soils with alpha value greater than about 5. Hope this helps. Thanks.--Rien.


Hi Rien
I am having some problems in using RETC to predict soil hydraulic parameters. The water retention data (below) are for a recent soil developed in dune sand. But I got alpha values 75 and 127 per meter if I use van Genuchten model. If I use Brooks & Corey model, I get alpha = 18 per meter and n = 0.6. Can n value less than one? In RETC manual, n values are 1.1-2.7.

Liping

meter water content weight
A -
0.51 0.273 1
1.02 0.209 1
2.041 0.192 1
4.082 0.178 1
10.204 0.166 1
153.061 0.103 1

C1-

0.51 0.08 1
1.02 0.048 1
2.041 0.041 1
4.082 0.035 1
10.204 0.03 1
153.061 0.027 1

C2 –

0.51 0.089 1
1.02 0.0545 1
2.041 0.051 1
4.082 0.045 1
10.204 0.0395 1
153.061 0.0225 1

Liping:
Your data do not have any resolution in the wet area to get a complete curve out of it. You need a point near saturation to fix the curve. The first point is at .51 m. Dune sands may already have lost most of their water. This is especially true for your 2nd and 3th data sets. Even the first data set has nothing to define the curve near saturation. When I ran it with RETC I got a saturated water content (WCS) of something like 0.7. Fixing WCS at 0.3 already gave more stable results (a good estimate for the saturated gravimetric water content?). In any case, you need additional measurements at or near near saturaiton to fix the curves.

The Brooks and Corey equation can get valuas of n less than 1 (their pose-size distribution index lambda). For your first exampel I got about 0.3. The n value for BC is about 1.0 less that the n value on our e2uations (i.e., nBC=nVG-1). Bottom line: get a few dune sand samples and measure at least WCS and maybe one or two more points near saturation. Thanks.--Rien van Genuchten

Hi Rien
I could estimate soil hydraulic parameters (Th_r, Th_s, alpha, n) either from water retention data (6-7 points) using RETC or from particle size distribution data (Sand, Silt, Clay, Bulkd, Th33, Th1500) using Rosseta. Rosetta also gives Ks. Which method will give more accurate results?

Liping

Liping:
Measured values generally should give the best results, provided they show good resolution between wet and dry. Your example data did not define the curve very well; hence I would have more confidence in Rosetta in this case (especially since Rosetta's accuracy is best for coearse-tectured soils). Still, you may want to compare the experimental data and the Rosetta results, and perhaps scale the Rosetta curve to force the retention function to go somehow through your measured data. Good luck.--Rien van G.

**ID = 261, DNAPL and LNAPL**

Dear all,

I am wondering if we can use hydrus 2D to model DNAPL and LNAPL transport through unsaturated and saturated zones.
Moayyad


Moayyad
No, you can not. Hydrus is a single phase model.
Jirka


## ID = 262, Animation result export ?

Is there any method to export animation results from HYDRUS2D?
I tried to use the animation result in Power Point Program, but I failed.
Anybody Knows?
Thank you
hasife@kigam.re.kr


Hi Hasife:
Unfortunately there is no method/function that could export Hydrus animation to a GIF file or other animation format. When I need to export Hydrus animation I always have to do following steps:
1/ Take screen images one by one with "Alt" + "Print Screen"
2/ In the "Paint" paste each image and save it for example as Picture_x.bmp where x=1,...,N (number of pictures)
3/ Use a program that can import all saved images and create an animated GIF (or other animation format). I use the Fireworks by Macromedia that can do it very easily.

We will take this problem into account during development of next Hydrus versions.
Mirek


## ID = 263, Constant pressure in simulation domain

Hello,
does anybody know if it is possible to subscribe a constant pressure somewhere in the simulation domain (not at the boundary nodes)? I want to simulate a lysimeter with suction cups. I know the pressure that is used at the suction cups and the location of the suction cups.

Best regards,
Heinke


Heinke,

The computational module does have such option, although the graphical interface does not support that at input. You need to do the following:

a) In the graphical interface, in the Boundary module, find the number of the node in which you want to specify this condition (Option->Display Options->Numbering of Mesh Nodes).
b) Save the input in the asci format (File->Transform to ASCII).
c) Close the project
d) In the input file Domain.dat (in the folder of a particular project. You can find the path in the project manager) change manualy the Code for a particular node to 1 (Constant pressure) and h to a particular pressure.
e)Open and run the project.

Jirka


## ID = 264, Microbial pathogens

Hi all,
could Hydrus 2d model the transport of microbial pathogens in the porous media with acceptable assumption.

Mo'ayyad


Yes, it could. See the following papers.

Schijven, J., and J. Šimùnek, Kinetic modeling of virus transport at field scale, J. of Contam. Hydrology, 55(1 2), 113 135, 2002.

Bradford, S. A., S. R. Yates, M. Bettehar, and J. Šimùnek, Physical factors affecting the transport and fate of colloids in saturated porous media, Water Resour. Res., 38(12), 1327, doi:10.1029/2002WR001340, 63.1-63.12, 2002.

Bradford, S. A., J. Šimùnek, M. Bettehar, M. Th. van Genuchten, and S. R. Yates, Modeling colloid attachment, straining, and exclusion in saturated porous media, Environ. Sci. & Technology, 37(10), 2242-2250, 2003.

## ID = 265, Precipitation

hi to all hydrusexperts!
I try to simulate flow through a constructed wetland which is fed every 6 hours with 16 liters of effluent per square meter.
i have problems specifying this with precipitation:

I have chosen cm and hours as units.
the feeding lasts only for one minute.
my domain is 100cm wide and 60 cm deep.

is this the correct variable boundary conditions?
time prec

0,0167 96 0 0 100 0 0
0,0334 0 0 0 100 0 0
6 0 0 0 100 0 0


or is it simply:
time prec

0,0167 1,6 0 0 100 0 0
0,0334 0 0 0 100 0 0
6 0 0 0 100 0 0

because in the help file it says:
Trans Potential transpiration rate [LT-1] (in absolute value).
that would mean the precipitation is in cm/h - what means "in absolute value"?
raimund


Raimund,
The first option you give is correct, since:
16 L/1 m2/1 min=16 dm3/1 m2/ 1 min=16000cm3/10000cm2/0.0167h=95.8 cm/h
Jirka


**ID = 266, Solute transport**

Dear Jirka,
I have a problem in running a scenario where I apply solid fertiliser in splits (60 kg first and then 170 kg/ha after irrigation)under furrow irrigation.
To apply solid fertiliser I assume that the fertiliser is already there in the soil - an initial condition (resident conc.). So I run the program two times, with 1/3 of the fertiliser first, irrigate (under head boundary condition). Then I use the output theta of this run to be the initial condition of the next run (no flux boundary). But for solute conc. I have to add the 2/3 of fertiliser conc.to those particular nodes where the fertiliser goes. It means that I manupulate the Conc1.txt file of the first run because I cannot manupulate the conc1.out file (which is binary). But the problem is that HYDRUS does not read the ASCII files correctly when I use that (ASCII file) as an input file for the next run (initial condition). I hope you understand my description of the problem. Do you have any suggestions?


Hi,
Hydrus runs the initial conditions from either the binary file Domain.in or an ascii file Domain.dat. When you import the initial condition from a previous run (you can import either pressure from h.out, water content from th.out, or concentration from conc1.out. You can still modify the initial condition after the import in the interface, or you can modify directly the Domain.dat file.

You can not modify conc1.out (binary), and no, you can not import the initial condition from an ascii conc1.txt. But I guess, the suggestion above should work for you.

Jirka


**ID = 267, Help for a couple of easy questions please !**

Hi
Probably real simple answers to these, but have left us stumped so any help would be appreciated – we are novices here !
Ran a simple 24 hr water flow experiment using hydrus for a number of soil types inputing specific particle size, water content and K data. Works fine for sandy soils but trips over for clays every time. Tried to find out where we are going wrong and only by changing the K value can we get meaningful data. However, we have to increase our experiment K value by an order of magnitude to get it to work. We end up using a K value for a clay (60%) that we might expect for a sandy loam which clearly isn't right. Any ideas, thoughts about what we might be doing wrong ? general advice ?

As a smaller issue, when you obtain a graph say theta V time, using the data grid editor you can see all the raw data. Can you not copy and paste this straight into excel ? it only seems to let you copy one cell at a time. Or are you stuck with hydrus's graphical interface ?


Van Genuchten-Mualem model is often not adequate for heavy-textured soil, such as clay, because of extremely sharp decrease of conductivity close to saturation. Choose instead VG-M model with 2 cm air entry. Often much better, and certainly more stable numerically. If you do not want to use this, then you will need to use finer spatial and temporal discretization to get a stable numerical solution.

Graphics - Hydrus graphics is not intended to be used for import of presented data (only graphs can be exported). Data are stored in ascii files in a folder of a particular project. There you can open them with any editor you want, including MS Excel.

Jirka


**ID = 268, Components of the anisotropy tensor**

Dear Jirka,
we want to simulate a solute transport experiment with anisotropy in the hydraulic conductivity. Right now I have the problem that I do not exactly understand what the two components (K1A; K2A)of the anisotropy tensor are describing and how to investigate them. We will make a experiment where we will observe the movement of a tracer plume in the soil, so that we will figure out the direction and speed of the movement. Can you explain to me the method to calculate the two components K1A and K2A and the angle w?

With best regards,
 Gunnar


Gunnar,
There is a detail information on p. 55 and 56 of the manual.
K1A and K2A are simply the scaling factors for the hydraulic conductivity in the x and z direction. Thus ratio of K1A and K2A is the ratio of conductivities in the horizontal and vertical direction. Angle is typically equal to zero, since the axis of the anisotropy tensor in most cases correspond with the x and z coordinate system. It is, however, possible to have this tensor position in different direction, such as for flow in a slope transect, where the anisotropy tensor can be at the same slope as the transect.

Jirka


## ID = 269, Reversal point in hysteresis

Dear HYDRUS users,
I have a question regarding a reversal point in hysteresis.
Is it correct that the reversal point is defined when the change in water content over a time step is reversed from the previous time step and greater than the tolerance?
Does it mean that the number of scanning curves that Hydrus will predict depends on the speficied tolerance?
Should I expect significantly different predictions if different tolerance are used?

Preecha


That is correct. There needs to be some minimal change, otherwise the program would switch branches of the retention curve (and conductivity) for every little numerical oscilation. The tolerance limit for hysteresis is the same as for the water content tolerance.

Jirka
## ID = 270, Slope saturation

Say, I have a rectangular hillslope (8deg) x-section where all boundaries are impermeable except for the surface rainfall boundary (atmospheric).Slope and soil depth along the entire slope length are uniform, and soils consist of two materials in the profile with one top is highly permeable and another bottom is impermeable. So, during prolonged steady rain, perched water table rises above interface and intersects the slope surface, where surface becomes rainfall as well as potential exfiltration boundary at downslope portion. I am defining this length of seepage boundary at downlslope portion along atmospheric boundary as a iterative matter to stop infiltration only that portion everytime and running Hydrus 2D(v-2.008)for each iteration and eventually to know the length of slope saturation and to compare with the observed.

But problem is that, if I consider this whole process occurs under steady state, then what should I do for initial conditions for each

iteration, should I transfer one iteration's final condition to next's initial, or I should run individual iteration for steady state as a whole process and compare with the observed and continuation of iteration until achieve agreed flow field profiles

within the variability of observed, i.e., similar initial conditions for each iteration, but just length of seepage boundary is different along the surface until agreement.

Uzzal,
As you well know, the standard version of Hydrus-2D can have only one type of boundary condition prescribed for any particular boundary node, i.e., free drainage, atmospheric, or seepage face. These boundary conditions can not dynamically change. However, in the past I have been working on the similar problem that you have, i.e., the dynamically overlapping atmospheric and seepage face boundary conditions. I could send you this version of the code. However, I'm at present traveling in Europe and will be back in Riverside on October 9. Send me an email to my own UCR address after this date (!) and I will send you the version that can handle this problem.
Jirka

**ID = 271, Slope saturation – addition**

Uzzal, 09/29/2004 :  12:17:40
I posted the previous msg before requesting advice or suggestion/s from your honor. I am sorry for that and for addition of the first msg.
Thank you very much for your kind cooperation.
Uzzal

**ID = 272, Automated Ascii Output conversion**

Hi Hydrus developers & users,
I was searching for a solution how to automate the output of Ascii files. So far I could not find a solution to let my program check the latest Hydrus results and let it change the input interactive and restart Hydrus again without user input. With userinput its no problem. But theres always the step when V.OUT, TH.OUT and H.OUT have to be converted to ASCII.
Uwe
The conversion of binary into the ascii files is done only after selecting command |Convert to ascii| and selecting which files. Interface does not convert these files automatically after the calculations are finish. Normally users never look at the actual numbers, and use only graphical tools, which are build into hydrus interface.

J.

**ID = 273, Continued simulation with hysteresis**

Dear all.

I've - as I hope a small - problem with continued simulations using hysteresis in retention curve.

I would like to simulate atmospheric long-term data over some years. In order to get results of water content or pressure head distributions of sufficient number I have to split the simulations in some different parts and to give the final condition of the last simulation as initial condition to the next one.

And here comes the problem. If I have chosen in "Soil Hydraulic Model" an initially calculation with e.g. "Drying Curve", the initial condition will be calculated new based on the distribution I have imported (water content or pressure head).

In short, if my calculation was located on a scanning curve at the end of the simulation, the inititial condition in the next simulation will be transformed on a primary curve I've chose as initial curve.

My questions are first, is there any opportunity to import the pair of parameters (water content and pressure head) in order to get the location on the scanning curve. And second, is it even possible to import the information if the calculation was on a drying or wetting process.

I hope there is someone giving me a good advice.
Alex


Alex,
You are right that Hydrus can be started only from the main drying or main wetting curve. We have realized recently that this is a problem and do plan to look at that with Heiner Stoffregen in the coming month. If we solve the problem, and write a version that will allow to start from the scaning curve, I will let you know.

Jirka


Dear Jirka.
Thank you very much for the quick response. I'm sure this upgrading could be of value for some tasks.
If there is new version of the model including this option I would be very interested in, of course.
Alex

**ID = 274, Evaluating the heavy metals accumulation in soil**

Hi
Can I use the Hydrus 2D in evaluating the heavy metals accumulation in an irrigated soil over a period of 100 years. The irrigation water is polluted with cadmium (also Zn and Pb).
In case of yes, what are the parameters should I be careful with?
thanks

It all depends on how complex is your system. Do you really need to use 2D? Perhaps 1D would be sufficient. Do you want to simulate cadmium as a sorbing chemical (with sorption described as instantaneous or kinetic process), that may be uptaken by plants. If that is sufficient for your application, then Hydrus is the way to go. However, if you want to consider more complex processes (e.g., mineral precipitation/dissolution) then you need more complex model (and I do not know any except our HP1 (HYDRUS-PHREEQC) that will be released in few weeks).

Jirka

Hi,
It all depends on how complex is your system. Do you really need to use 2D? Perhaps 1D would be sufficient. Do you want to simulate cadmium as a sorbing chemical (with sorption described as instantaneous or kinetic process), that may be uptaken by plants. If that is sufficient for your application, then Hydrus is the way to go. However, if you want to consider more complex processes (e.g., mineral precipitation/dissolution) then you need more complex model (and I do not know any except our HP1 (HYDRUS-PHREEQC) that will be released in few weeks).

Jirka

If the pollutant is in the irrigation water (low quality water) i.e. the pollutant enters the system only during irrigation periods. How I can simulate this situation?

Hello,
You could probably do that by introducing a time-dependant boundary conditions. In Variable Boundary Condition of Pre-Processing Menu :
-Irrigation : rGWL or GWL, it depends on the kind of your irrigation I think (flux or head)
-Associated solutes : ConcX (cvalueX) is the time-dependent solute concentration [ML-3] for solute X and is used for nodes with prescribed time variable BC (atmospheric BC, variable head/flux BC)
In the editor, you may choose which water and solute boundary condition is assigned to a specific part of your domain (pointer). See also FAQ 29.

I have one more question, maybe for Jirka, about time variable solute (NO3) BC : is it possible to use this option to simulate net nitrification in the time (I did laboratory incubation with and without crop residues) or is there an other easier way?

Julie

Julie,
The way I understand this, nitrification is a process of converting amonium NH4 to NO2 and futher to NO3. Denitrification is further conversion from NO3 to N2 (gaseous). Each of these processes can be simulated in HYDRUS as a first order

process. We actually have one example dealing with this. Test3 "Solute transport with nitrification chain". See the manual.

Jirka


Thank you for you reply and advice.
In fact, I was conscious that this process could be simulated as a first order process. But in my experimentation, there is a big flush of nitrification after application of chemical and the latter kinetic of nitrification is different at the beggining and at the end of the study. I was wondering if the utilisation of "cvalueX" (the time-dependent solute concentration) was not better to simulate "my" data (Am I clear ?).

The soil I work with is a variable charges soil (andosol) and there is a strong pH effect (on mineralization/nitrification but also on the adsorption). Have you ever worked on this kind of phenomena with Hydrus 2D? Will Hydrus/PhreeQC take into account the effect of the pH on biological and chemical processes?

In the exemple 4, I am confusing with units : in test4, it is in days and in the manual, it is printed in hours.

Julie


Julie,
I can see only one way how you can change the rates of reactions in Hydrus with time. Since Hydrus considers the effects of temperature on the transport and reaction parameters, you may simulate simultaneously heat transport, and change the temperature of the system (by using time variable temperature boundary conditions, and high thermal conductivity), so that it affects your reaction rates.

Coupled model HP1 (Hydrus1D-Phreeqc) can consider effect of pH on chemical and biological reaction (see our example of pH-dependent cation exchange in "Jacques, D., J. Šimùnek, D. Mallants, and M. Th. van Genuchten, The HYDRUS-PHREEQC multicomponent transport model for variably-saturated porous media: Code verification and application, MODFLOW and More 2003: Understanding through Modeling, Conference Proceedings, ed. E. Poeter, Ch. Zheng, M. Hill, and J. Doherty, International Ground Water Modeling Center, Colorado School of Mines, 23-27, 2003."). During my recent visit of Mol, Belgium, we finished the first version of HP1 and we should release it very soon.

Units in Test4 should be days.
Jirka


Julie,
I can see only one way how you can change the rates of reactions in Hydrus with time. Since Hydrus considers the effects of temperature on the transport and reaction parameters, you may simulate simultaneously heat transport, and change the

temperature of the system (by using time variable temperature boundary conditions, and high thermal conductivity), so that it affects your reaction rates.

Coupled model HP1 (Hydrus1D-Phreeqc) can consider effect of pH on chemical and biological reaction (see our example of pH-dependent cation exchange in "Jacques, D., J. Šimùnek, D. Mallants, and M. Th. van Genuchten, The HYDRUS-PHREEQC multicomponent transport model for variably-saturated porous media: Code verification and application, MODFLOW and More 2003: Understanding through Modeling, Conference Proceedings, ed. E. Poeter, Ch. Zheng, M. Hill, and J. Doherty, International Ground Water Modeling Center, Colorado School of Mines, 23-27, 2003."). During my recent visit of Mol, Belgium, we finished the first version of HP1 and we should release it very soon.

Units in Test4 should be days.

Jirka
I have not received a clear answer for my question, what are the parameters I should be careful with when I evaluate an irrigated soil using water polluted with cadmium (properties or coefficients effect cadmium movement in the soil)

Ahmed

## ID = 275, Inverse Convergence

All,
I am attempting to estimate hydraulic parameters using the inverse solution method and the numerical solution is not converging. Would this be the result of a poor initial estimate of the soil's hydraulic parameters, the time discretization parameters, the iteration criteria parameters, or none of the above?

Always try to run the problem first in a direct mode, i.e., inverse option with zero inverse iteration. Then after reviewing the first output, you can see whether all your input data were correct and whether your inverse problem does make sense.

J.

Thanks. I see exactly what you mean and that has helped a lot. I think that I am setting up the problem incorrectly. My data are ponding heights above an infiltrative surface after an initial input of water in a trench. Would you suggest that I make my upper boundary condition an atmospheric BC with a single pulse of precipitation at the beginning of the simulation or a variable pressure head BC with the same data as I am using in the inverse solution screen?

Make it an "atmospheric BC with surface layer" and specify the initial height of standing water as positive pressure in the top node (in the initial condition). Hydrus will take care of calculating how long it will take for the water to fully infiltrate into the soil profile.

J.

OK, I've set the initial conditions as you suggested and it works great, but I'm still having a problem with convergence. The rsquare of my initial estimate of the hydraulic parameters is over 0.98 and the inverse function is still not working. Is the rsquare the indicator I should be looking at to know whether my estimate is close enough to the actual values or are there other indicators? If it is rsquare, how high does it need to be before I can expect results from the inverse function?

Thanks again.

Look at rsquare, look visually at the fit, and look at the confidence intervals of optimized parameters.
J.

I have looked visually at the fit and at the confidence intervals. The simulation under-approximates the infiltration rate near the beginning of the tests and over-approximates it later. The confidence intervals have a huge spread for most if not all parameters. Is there a certain visual fit or confidence interval that I am shooting for before the inverse function will work? I am currently attempting to solve for Qr, Qs, alpha, n, and Ks and have reviewed how varying each of them should affect the infiltration curve. Is that too many parameters to solve for at once? I have tried fixing Qr and Qs and just solving for the other three but again to no avail. Any suggestions?
Thanks

**ID = 276, Stanmod Inverse Modeling - fit to averaged conc.**

Dear list,
I want to fit parameters to time averaged concentrations from sample volumes accumulated over periods of about 30 days using Stanmod. As far as I understand, the current least square optimisation routine minimizes residuals with regard to concentrations measured at a point in time.
A work around could be to take the averaged conc. as the "point in time conc." in the middle of the sampling interval. However, with a conc. peak within the interval, there could be quite a difference between those two conc.
It seems questionable if the fit is close to the "true" best fit that actually would minimize residuals with regard to integral conc., doesn't it? Or is it okay to do this?
If not, does anyone have a better recommendation on how to deal with this issue using the current version of Stanmod?

Sven
We once investigated the effect of the time interval on time-averaged
concentrations using analytical expressions (see Fig. 4 in Leij and
Toride, WRR, 31 1713-1724, 1995). The problem can occur only in extreme

cases with large sampling intervals and low dispersion coefficients. Generally speaking, I feel other unexpected errors in experiments including the validity of the CDE would be greater than the sampling interval effect. Please firstly use CXTFIT as it is. We had better carefully investigate the results before applying complicated solutions. If any further questions, feel free to contact me. Regards,

Nobuo


## ID = 277, Notebook

I have a question about registration.
I have single user license for HYDRUS2D 2.102 installed on my computer. But have problem now if I can acquire license to install Hydrus to my notebook to.


Hi,
I think you are eligible to get one more license for your notebook. Please contact IGWMC (igwmc@mines.edu).
Mirek


## ID = 278, Water content and Solute transport

I am Khalil Azhdary a Ph.D student in IARI New Delhi India. I am able to run the Hydrus2D with given inputs but still I have some problems in given the boundry flux at certain nodes.
I am simulating water and solut transport under drip fertigation. The problem involves appling irrigation water through drip for 2 hr with 4 lit/hr discharge and then no application of water for next 46 hr. The next irrigation is given ofter 48 hr with the same irrigation schedule. Kindly suggest me how to give the input of 4 lit/hr for 2hr and 0,0,0,...... for remaining 46hr.

Khalil


Time-dependent boundary conditions
Record no Time Flux
1 2 xxx
2 48 0
3 50 xxx
4 100 0

xxx stands for your flux. In the abve example there is irrigation between 0 and 2 hours, and again between 48 and 50 hours. There is no irrigation in between.

Jirka

Dear Simunek

Thanks for your replies and I think your suggestions are correct but still my problem is not solved. I am simulating water and nitrogen transport under surface point source (Drip Fertigation) and I want to compare the observation value of water content and nitrogen movement with simulation value. My queries are as follows:

1. In case of water content:
a. How can I select the iteration criteria parameters in my case?
like lower and upper time step multiplication factor, lower and upper limit of the tension interval. And also why it is running with pressure head and not running with water content? ( only in the iteration criteria)

b.In my case, where should I give flux boundary? Whether in Nodal Recharge or by some other option?

c. As I have metioned earlier, I would like to do the simulation with 4 litre/ hour flux applied only for 2 hours and then no flux for remaining 46 hours. How should I mangage this flux boundary. I want do the simulation for 48 hours.

2. In case of solute transport:
I am applying solute along with the water through drip. Water is applied for 2 hours whereas, solute i.e fertilizer is applied for initial 30 minute. For remaining period no fertilizer is applied.

a. How should I manage this boundary?
b. What should be the unit of concentration?
c. how should I give the initial condition?
d. What is pulse duration and how it it is define in the model?
e. I am defining The initial condition i.e intitial concentration of N in kg/ha and the the solute flux in the model is given in kg/litre or gram/litre. How to manage this problem?

Hi,
I can not really go through every detail of the program. I'm emailing you an example of the drip irrigation run that has two periods of irrigation, each with a shorter fertigation period. Most, if not all, of your questions should be asnwered by carefully styding this example.

J.

Dr. Simunek

Thanks for your replies, it was very helpfull to me.But in sourface drip example, there is no value for variable flux and concentration in the example sent by you. In my case emitter discharge is 4lit/hour. Is it not necessary to specify value of point source in boundary condition? Can I add one more column in Time dependent Boundary condition editor to enter flux?

Where can I give the concentration value?

Azhdary

## ID = 279, Problems using Brooks and Corey

HI all i am try to use Hydrus to estimate drying due to ET and the model i am using is Brooks and Corey model for soil retention curve with Qr= 0.02 , Qs = 0.385, alpha = 0.067, n = 0.95 , Ks = 9.5 l = 1
units are cm and hrs.
The model gives divsion by zero error when i start with water table at 15 cm below land surface, if i start below that then then it gives convergence problem, i am simulating a 500 cm colum with simple surface evaporation which works fine with Van Genutchen model.But as soon as i change to BC model i am getting this problem.

nirjhar
Disable the interpolation tables in the "Iteration Criteria" dialog window, by specifying both limits of the interpolation table equal to zero.
J.

The simulation now seems to be working and converging.
However can u suggest that if i start the water table at a depth less than the capillary fringe (=ha for brooks and corey) then it dosent converge. It this because of iteration criteria or an artifact of the model, because in VG model u can start at any depth as small as 0.1 cm .
nirjhar

## ID = 282, Geotechnical Laboratories

We have the opportunity to collect soil samples for analysis of VG soil input parameters. This application will be compared to HELP simulations, so we will also be analyzing for moisture content at 33 and 1500 KPa.
Can anybody recommend a geotechnical laboratory that can perform these analyses.
Thanks.

I would recommend the following lab:
Daniel B. Stephens & Associates, Inc.
phone: 505/822-9400
fax: 505/822-8877
http://www.dbstephens.com
Jirka

## ID = 283, Impact of septictank effluent disposal on GW quality

Hi Gurus,

I try to simulate the accumulative impact of soak hole-disposal of septic tank effluent on groundwater quality (NO3 and faecal coliform) through a stony soil profile and coarse gravel aquifer. In a 4-km area, there are many soak holes (about 4 m deep and 1 m diameter) and lots of drinking water wells (20-40 m deep).

The size of model domain will be 4000 m wide and 100 m deep. Compared to the size of the model domain, sockholes and wells are very small. I wish to generate a mesh that could be appropriate for Hydrus calculations. My questions are:

1. Should sockholes and wells are best inserted as circles in the model domain at the depths where their bottoms are?

2. For the boundary points: what would be the desirable numbers of points on the outside curve and around the sockholes and wells, respectively?

3. What would be the desirable density of boundary points (left and right) for outside curve and around sockholes and wells?

4. I am thinking to use atmosphere upper BC, no flux lower BC, and constant pressure heads for the L and R BCs. For the inner boundaries, I will use constant flux for sockholes and wells. Does this sound right?

5. Whether it would be easier to split the problem with two models? The first model has a depth of 4 m with free drainage at lower BC. The second model gets rid of soil layers above 4 m and thus the sockholes can be assigned at the top boundary as constant flux upper BCs. The areas between sockholes are atmosphere upper BCs using the cumulative fluxes at the bottom of the first model as "rainfall". There will have then no evaporation in the second model. I hope this approach may avoid the irregular mesh problem that I question in 1-3. How do you think?

6. The sockholes are developed at 4 m deep. Should I still consider denitrification? How could HYDRUS model denitrification?

Wish to hear from you. Thanks a lot in advance.
Liping


No reply yet to this question?


### ID = 284, Stream tube model - standard deviation of v and D

When applying the stream tube model to the example of Figure 7.12 (Jury et. al. 1982) one obtains - according to the CXTFIT Manual and the example output file - $<v>=30.5$ mm d^-1, $<D>$ =2.5 mm^2 d^-1, sigma_v =0.8 mm d^-1 and sigma _D = 0.8 mm^2 d^-1.

Are sigma_v and sigma _D as given here, the standard deviation of the log-transformed variable? According to the variable list of the manual this should be the case. If they are standard deviations of the log-transformed variable, is it correct in

this case to quote them with units? This may give the impression that they are given for the cartesian space, doesn't it?

Sven
Thanks for your comments.

>Are sigma_v and sigma _D as given here, the standard deviation of the
>log-transformed variable?

Yes, you are write. Sigma is for the log-normal distribution. The mean and standard deviation for the Cartesian space are given by (A.34) and (A.35) in Soil Physics 6th edition of Bill Jury (or (8.34) and (8.35) in 5th edition).

>If they are standard deviations of the log-transformed variable, is it
>correct in this case to quote them with units?

I should drop out these units for sigma for the log-transformed variable.
Bill Jury did not give units in examples in his book.

Nobuo

**ID = 285, Problem using hydrus2D !**

I used Hydrus2D for a rectangular domain (30 cm width and 200 cm height) by linking lines to four corner points and using triangular meshes. The bottom boundary is h=0 and the upper boundary is h=0. The initial condition correspond to a linear decrease of h with depth from top at -1000cm to the bottom at h=0 . The soil is that of clay using VG Mualem model for soil hydraulic properties. I realized many test cases by changing initial, minimal and maximal time steps, values of the lower and upper limits of the time step multiplication factors, lower and upper limits of the tension interval, ... but I couldn't run hydrus2D at the first time step. Because of the machine speed, the printing on the screen can not be viewed netly, I can not catch where the problem comes from. The only thing I observed rapidly from the screen is that there is an error once reading printing times " error, invalid number " ?

could you help me to solve this problem !

SAADI Z.

SAADI,
I would expect that the problem is with your mesh. You can not have points in the corners of the domain, that you then connect with lines. That results in dupicative nodes in the corner. You just need to use lines. Individual points can be in the transport domain, but can never be at boundary curves.

Jirka

**ID = 286, Can this be solved using Hydrus_2D**

I would like to model the following using Hydrus_2D. Is it possible?
\The aquifer is to be modelled in vertical section with depth 40m extending 250m horizontally. Axisymmetric flow is to be implemented to a single subsurface point simulating a well. The program has to model numerically solute transport in both saturated and unsaturated media taking into account active processes including non-equilibrium sorption and biodegradation.

The main aquifer is assumed to be stratified, divided into three layers – the upper layer (0-10m); middle layer (10-20m) and lower layer (20-40m). Each layer or stratum is assigned appropriate soil material and hydraulic properties based on the generic lithology.

The cases to be considered include the effects of recharge due to rainfall, floods etc., seasonality, root water uptake, potential evapo-transpiration, the non-equilibrium adsorption of solute onto solid phase and microbial reduction on the migration of solute from strata close to the ground surface to the well screen at depth due to pumping.

The main processes that are to be looked at are as follows:

1. oxidation and reduction of the solute sorbed on the solid phase due to the change in water table.
2. what happens to the solute if there is no change in water table?
3. How long will it take for the solute to reach from surface to the well?
The program has to be run for a period of 100 years.

valliyappan.t


Valliyappant:
In principle this should be possible, provided you can capture the oxidation-reduction processes with a (equilibrium of nonequilibrium) Freundlich and/or Langmuir isotherm, perhaps combined somehow with the first- and zero-order terms.

The axisymmetric domain is very large (especially deep). Simulating the problem for 100 years may require some time-averaging of the atmospheric conditions (rainfall, ET), e.g., into monthly (or even yearly?) values. I would assume that you have no significant regional flow to interfere with the assumption of axisymmetry. Thanks.—

Rien van G.


**ID = 287, Seepage and Free drainage**

Hi

For a simple domain that consists of infiltration (constant flux) for top boundary, simulating the bottom as seepage face once and as a free drainage another time. Using a seepage face at the bottom, the simulation reached steady state, while using a free drainage boundary at the bottom, water started to store on top of the bottom face and started to accumulate upward.

With the definitions given to seepage face and free drainage, wouldn't you expect the opposite. In other terms, shouldn't the free drainage dissipate more volumes of water?

Mazen

Mizo:
Yes, free drainage (zero pressure head gradient) should remove the water more quickly from the bottom than a seepage face. Unless perhaps when you have a layered profile in which the conductivity is low near the bottom boundary. This because the bottom flux then cannot exceed the hydraulic conductivity there, while for a seepage face a large positive pressure can develop to push the water out at a higher rate. Does that make sense? Thanks.--Rien

Hi Rien
So, do you think for that specific case that you mentioned (one with seepage face at bottom), water storage can build up?
Do you think a seepage face is a good representation of a GDL? or would you rather use free drainage boundary for that?

Mazen

**ID = 288, Solution oscillations**

I'm simulating solute breakthrough from a field with a ditch on one side with constant rainfall over a period of 30 years. After about 3 years I get oscillations in my water and solute flux solutions. First around $2.25 \times 10^{-4}$ and after a few more years around $2.25 \times 10^{-3}$. My mesh is quite fine and simulations with varying precipitation are oke. What is wrong?
Lies
Lies,
I can not see any reasons why you should get oscillations in the water flow solution.
I do see possible reasons for oscillation solute transport solutions. Oscillatory behavior of solute transport numerical solution and way how to overcome it are dealt with in chapter 6.4.6 of the manual. Apparently, convective transport dominates dispersive transport. You need to take some stabilizing measures discussed in chapter 6.4.6, i.e., upstream weighting, artificial dispersion, using stability index to determine time step, or increase dispersivity. Check the Peclet numbers in the output. If they are larger than 5 then numerical instabilities are to be expected.
Jirka

Dear Jirka,
Thanks for the answer, but that is not the problem. I found out that the problem is the use of cumulative values in the solute1.out and Cum_Q.out files. Because the accumulated values get very large, there is apparently a limitation in the accuracy of the presented numbers. Which in my case looked like oscillations.
Lies

## ID = 289, Hydrus break down

Hello,
I'm starting learning to use Hydrus 2D and I use a simple case of infiltration of water. (initial conditions in water content, boudary conditions of constant pressure on the top and free drainage on the bottom. the discretization is rectangular and the hydraulic model is of Brook and Corey). The simulation works and seems ok, but the I can't view the observation nodes (they are three and placed near the top, in the middle and on the bottom). When I try to look at the results for the observation nodes, the program crashes and stops. (the other results are ok) What is wrong?

Hi Sales,
this is really strange. Can you display results in observation nodes in other projects (demo projects)? If yes, please send me the project including results (the file "Name".h2d + directory "Name"). Use Winzip if you can.

Regards Mirek

## ID = 290, Reading a new object from a file ?

Hello,
I'm trying to import a file with 2 columns (100 rows) corresponding to the values X,Z of a boundary curve but I couldn't read it using the Icon XY "new object from a file" (polyline or spline ?)

How can I do this ? is there any specified format for the file to be imported ?

Z. SAADI
Saadi:
the format is very simple and is described in the help. Example:

4
1.1 3.400
1.4 3.566e-2
1.200e-1 100.
0.00012 134.

The first number is number of rows and then there are always two numbers (X and Y/Z) on each line in "free format" (separated by one or more spaces).

**ID = 291, Boundary Types**

Dear all:
I'm simulating a 150cm landfill cover system, 10% slope. I applyed a no-flux boundary at the upstream (right-side) face and a seepage boundary at the downstream (left-side) face. The purpose of including a seepage boundary is to simulate the effect of vertical drainage pipes.
The problem is that, in real life applications, drainage pipes are (commonly) about 50 meters appart, which leads to a lower seepage flux than predicted by the 150cm long model. So I'm left with to alternatives:
1- increase the length of the modeled system to 50 meters, which may necessitate an exetremely long running time.
2- change the boundary condition, but I'm not sure which type is the closest to reality.

Any suggestions???
Sofia Ghanimeh

**ID = 292, Reading a new object from a file !**

Mirek,
Is it possible to have more than 5000 points on the boundary curve if we want to mesh a transport domain with a boundary curve with a number of points (polylines) from the xy file which exceeds 5000 pts ? I'm using HYDRUS2D version 2.008.

Saadi:
If you have N>1000 points on boundaries then your FE-mesh will have millions of nodes (N*N). Calculation would be very slow and I don't believe you could get results in an acceptable time. Why do you need so many points on boundaries?
Mirek

Mirek:
I need many points on the boundaries because lines on the boundary curve are linked by more than 5000 points (especially 6580 purple points). When meshing the transport domain the number of points that can be created by hydrus2D (green points around purple points) on the boundary can not exceed 5000 points.
Z.SAADI

Saadi:
I don't know how complex is you domain but I don't believe that you really need 5000 points to describe its boundaries. You have to simplify it otherwise if you have more than 5000 boundary points then your FE-Mesh will have about 25000000 nodes and then you should contact a super-computer center and ask them to solve your problem. There is no chance to calculate such simulation on a PC.

Send me the file with XY coordinates of your boundary curves and perhaps I'll find a way how to simplify the boundary so that the mesh had a reasonable size (5000-30000 nodes).
Mirek


## ID = 293, Source code?

Since HYDRUS1D is public domain, I was wondering if it is at all possible to get hold of the source code. If not, is there any other way of disabling the requirement of pressing ENTER to exit the program? I'd like to use HYDRUS1D with another model, and this feature is making it very difficult.

The source code is not available but you can solve the problem by running H1D from a batch file:

1. See the post from IJsbrand:
http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=51
or
2. FAQ 27
http://www.pc-progress.cz/Fr_Services_Hydrus_FAQ.htm

Regards Mirek


## ID = 294, Vertical flow Vs Axisymmetric flow

I have modelled an aquifer with the following dimensions 10000 cm (Length) and 4000cm (Depth). I am using both the flow and solute transport equation. I have used an atmosphreic boundary on the top (all through the 10000 cm; which is to include precipitation, evaporation and solute transport. I have used a free drainage(which acts as a well); which is located in the middle (at 1900 cm depth, 3 nodes selected) of the left-hand side boundary. But not in the middle of the aquifer. I have used a linear distribution with depth (-100cm top and 300cm bottom) for the initial flow condition.

I ran the model for 365 days using vertical flow. I got reasonable results.

But keeping all the data's same and if i run the model using axisymmetric flow, am not able to get reasonable results. The results are totally different and unexpected.

-----------------------------------------/

2. Keeping everything same as the above and changing the linear distribution with depth to (-1000cm top and 3000cm bottom) for the initial flow condition and I ran the model for vertical flow.

The model gives me an error message after a short period of time, then stops the simulation.

valliyappan.t

**ID = 295, Quantification of dispersion**

Hello,
Is it possible to quantify the total dispersion generated by Hydrus2D, including the numerical dispersion resulting from the space and time discretisation? Especially for cases when the Peclet number is largrer than 1.
Lies


Lies,
HYDRUS should produce very little artificial dispersion with any realistic discretization. This unless you select upstream weighting or GFE with artificial dispersion to avoid oscillations if the elements are relatively large. Oscillations are controlled by the Courant and grid Peclet numbers as explained in section 6.4.4 of the manual). To have some fun and see what happens, run HYDRUS-1D a few times for different nodal spacings (solute transport during steady flow) and analyze the results (inverse analysis) with an analytical solution (e.g., CFITM within STANMOD). Thanks.--Rien van G.


**ID = 296, Negative pressure head**

Hi,
I want know why I got negative pressure head. I've got hydrostatic conditions, h=o at the top and h=1 (depth of my column) at the bottom with linear distribution. I try to simulate an constant pressure head injection (h=0.01) at the top of my column. When I have a look to my results I've got negative pressure head? Why?

Jeff


**ID = 297, Hydrus 2D density driven simulation capabilities**

Hi
I have to simulate landfill leachate migration in groundwater. The leachate seems to sink due to his density higher than groundwater density (density driven contaminant migration).
Is Hydrus 2D able to simulate this density driven contaminant movement?
Thanks


No, Hydrus does not consider density dependent flow and transport.
J.

**ID = 298, Maximum limit for inverse observations**

Is there a maximum number of observations one can enter for inverse solutions? I have a model I was trying to run on hourly time step for about 2 years, and approximately 14,000 observations for the inverse solution. When trying to run the model, I got the error
'Dimension in NOB is exceeded, stop!'
I'm sure I had the number of observations declared correctly, and when I used the same model but turned off the inverse solution portion, it ran fine. Therefore, this lead me to believe there might be a maximum number of observations that cannot be exceeded. If so, what is the maximum number, and can it be changed somehow in the inputs? Alternately, is there some other potential reason for this error?
Thanks.

Kelly,
This is indeed "Number of observations". I have been changing this dimensions in the past, but for the last three years I have there maximum of 7000 data points. Frankly, that seems way too many to me. You need to filter our some data points. You do not want to have so many data points since that significantly slows down the inversion.

Jirka

**ID = 299, Hydrus 2D book**

I am a Hydrus 2D user. I have got the old Hydrus 2D manual. I
came to know from the website www.hydrus2d.com, the new manual will
be more informative.
So would like to order one copy of the book. I did e-mail to the address hydrus@optusnet.com.au
Is there any online order and payment process available? So that i can receive the book by post as soon as possible.

valliyappan.t

I have copied your posting to the publisher that you urgently need the book. We are not involved in distribution of the book and thus can not help you with that.
Jirka

**ID = 300, Import initial concentration**

Dear HYDRUS Users
I am trying to import the end results from a first simulation into a second. The import of the pressure/water content works fine, but I have a problems with the import of the concentration. I do the same as I did for the pressure, but the imported solute volume

is always much higher then it should be (I compare the ConcVol in the Mass Balance Information). I tried it several times but couldn't find a way to solve the problem. I am working with the version 2.008 of HYDRUS 2D.
Martin Frey


Martin,
I have checked this on Plume examples from the Direct1 workspace and the balances at the end of the simulation for Plume3 and at the beginning of Plume4 are exactly the same (rerun it and compare with your version of Hydrus). You need to make sure that you export both initial pressure heads (or water contents) and concentrations (and nonequilibrium concentrations when nonequilibrium is considered).

The only reason for differences that I can see is when you additionally change something in initial conditions. This can be done, for example, also by changing boundary conditions for water flow. The constant head BCs are stored in the initial conditions (since these values are constant with time at the boundary, there is no need to store them elsewhere).

Jirka


Make sure to have the correct pressure head (and hence water content) distribution applicable to the same time level as you imported the concentraitons. Even if the concentrations are correct, but the pressure heads are not for the same time level, then the total amount of mass in the systems will be incorrect (unless everyhing is saturated). --Rien van G.


Dear Jirka
Thanks a lot for your help. My problem is that I also change the solute transport boundary conditions from the third type to the volatile type, if I don't do that then the import of the initial concentration works fine.
Is such a change in the boundary conditions not possible? This change is also the reason why a divided the simulation into two parts. I thought that this change should not effect the initial concentration.

Martin
Changing the boundary condition from the third-type to the volatile type should have no effect on the mass in the domain (and has now effect on initial imported concentrations). I did such example here and the code worked fine.
If, however, you change distribution coefficients, such as Kd and/or Henry constant, that would obviously effect mass in the domain, since what you import is only the liquid concentration and concentration in other phases is adjusted accordingly.

Jirka


**ID = 301, Units of Henry's law constant**

Dear colleagues
In Hydrus help, the Henry's law constant has the units of [M-1L3] so if I use meter and mg in my model, it would be m3/mg. I have found some literature values of Henry's law constant, but they are expressed as Pa m3/mol. I could convert mol to mg by dividing with the molecular weight but I don't know how to convert Pa m3 to m3.
But when I checked the 'list of variables' in hydrus 1D/2D manuals, Henry's law constant is dimensionless.
I am a bit confused and would appreciate very much if I could hear some comments on this.

Liping


Liping,
Units of the (Henry's) constant as entered at input is dimensionless, since it relates the gas concentration, which is in units of mass/volume to the liquid concentration, which is also in units of mass/volume. Thus this (Henry's law) constant must be dimentionless.
See also equation 3.4 of the manual and the text below on how this constant is related to the Henry's constant from various thermodynamic tables.

Jirka


## ID = 302, Loss through volatilisation

I am simulating pesticide leaching through soils using Hydrus-1D. Some pesticides are volatile and I wish to work out the percentage of loss via volatilisation. I have looked the example in Direct\Volatile and noticed that the column CVTop in solute1.out contains some negative values, which indicates volatile loss. But I am still unclear how to calculate the total volatile loss.

Liping


Liping,
Hydrus calculates and reports only the total flux across the boundary. Thus in the case of the surface, it sums up fluxes into the profile with infiltrating water (negative, since downwards), as well as fluxes out of the profile by gas diffusion (positive, since upward). Thus it should be easy to separate them just based on the direction. The only problem will be during the infiltration, when surface is not fully ponded, since at that moment there are both fluxes acting simultaneously, i.e., downward infiltration with water and upward gas diffusion. However, even in such case it should be possible to separate those fluxes since you should know how much goes downward (specified as BC). While in example Volatile, I applied solute in many surface fluxes, in real case there would probably be only one application, and thus separation of different fluxes should be easy and simple.

All this information can be found in the Solute1.out file.
Jirka

Hi Jirka

Many thanks for your comments and help. It is much appreciated.

After having read your comments, I tried the following calculations.

Total loss of solute flux via volatilisation = total solute flux input – total solute flux cross the soil surface

A. total solute flux input (M/L2/T) = sum of (rainfall*rainfall concentration/time duration)

B. total solute flux cross the soil surface (M/L2/T) = sum of (CVTop (in Solute1.out) * DT)

If I don't times the DT (time intervals between two time steps) with CVTop, it will give me a greater B value than that of A for the example "volatile", which is not realistic.

I then applied this approach to my real problem. A pesticide was applied at only one application at day 310 as CTop in atmosph.in. At that day, it has only rainfall value with a zero value of rSoil and rRoot. I assigned the Henry's constant of 63.6 (dimensionless) for all 6 soil layers and allowed no degradation. But I got no any volatile loss, i.e. A= B. I wonder why this could happen?

After the application day,

tAtm Prec rSoil

……………………

310 0.005 0
311 0.002 0.0024
312 0.002 0.0021
313 0.0114 0.0021
314 0.0072 0.0032
315 0.0048 0.0035
316 0.0006 0.0032
317 0 0.0052
318 0.0012 0.005
319 0 0.004
320 0.0062 0.0027
321 0.0058 0.0012

…………

all rRoot = 0

I have sent my files to your email address as I don't know how to attach files in this website.

Thank you very much again indeed for your kind help.

Liping


Liping,

I have looked at your problem. There are several problems with your input data.

a) You specify the gaseous diffusion coefficient (Diffus.G) equal to zero. Hydrus considers only gaseous diffusion as the transport mechanism in the gas phase. The volatilization into the atmosphere is by gas diffusion. Thus you need to specify this value otherwise you will not get any flux to the atmosphere.

b) Your retardation coefficient (as a result of Kd and bulk density) is enourmosly large. R=1+Bulk.Density*Kd/theta=1+1.e+9*1e+9/0.5=1e+18. Yo must have units wrong. This solute clearly can not move in any phase.

Once I changed these values (ro, Kd, and Dg) I got reasonable results that I would expect. If I'm right here, next time we meet you own me a beer. This note is for other people to see that my services are not completely free. :-)
Jirka


Jirka
Thank you very much for pointing out two obvious mistakes in my input file. I did not realised that I had accidentally copied the bulk density values as Kd values (Kd values should be in the orders of E-9 and E-10 in my model not in the order of E9!). After I have fixed these two mistakes. The results are reasonable and I got 88% of trifluralin-applied loss through volatilisation. For sure, I own you 10 bottles of beer or Whisky and I promise to pay you back when I see you next March!!!

One more question: what are reasonable values for Diffus.G and Difw for silt loam (0-39 cm depth) and silty clay loam (39-120 cm depth)? Is there any way to estimate these values for different soil types? How about for coastal sand?

Liping


Liping,
I believe that both of these two values, i.e., diffusion coefficients in liquid and gaseous phases (in pure phases), are independent of the soil. You can find them in various tables. The effect of soils comes in through the tortuosity factor, and in Hydrus I use Millington-Quirk model (which people may argue about).

Jirka


**ID = 303, How to modify and delete a question just posted**

Jirka
Thanks a lot for all your help. I really appreciate that. I own you another beer!

I wonder whether there is a way to modify and delete a question that I have just posted before I have heard from someone.

I also wonder whether I could get an automatic notice every time after I have posted a question so that I know it has been usefully posted. I only get automatic notices for someone else's questions.

Liping


Liping:
you can edit your postings but you can not delete them. There is a simple rule: everyone can only edit his own contributions.

How to edit:
1. Open the topic with your question
2. You should be logged-on and then you should see the "edit" icon on the top margin of the window with your message.
3. Click the icon to edit your text.

If you want to delete a topic just let me know and I'll do it (deleting requires administrator rights).
E-mail notifications of authors: I think that there is no option that would allow this. But if you really want it you can try to subscribe once more (under a different name but same e-mail address) and then it will work (perhaps).

Mirek


## ID = 304, Root solute uptake

Hi Jirka
Thanks for your last replies. I am trying to estimate cumulative root solute uptake by onion crop. I have 2 specific quarries:
1. while given the values under root uptake distribution of boundary codition editor, in which unit I should enter the value?
2. When I run the Hydrus-2D by given the values of root solute uptake boundary in fraction I am not getting any plot in cumulative root solute uptake of solute fluxes in post-processing section. why?


Khalil
1) This is a relative distribution of roots over the domain. Hydrus then integrates and normalizes your input. Thus select large values in regions where large uptake is, and vice versa.
2) You need to specify the value cRoot (the maximum allowed concentration uptaken by plant roots) in the "Reaction Parameters for Solute" dialog. If this value is zero then roots take up only water.

Jirka


Dear Jirka
Thanks for your suggestion it was very useful to me after given the cRoot in Reaction Parameters for solute I am getting result. However I have one more query.

In the drip example sent by you initial concentration for solute has been taken zero. In my case I am giving value of initial concentration equivalent to the initial value of the nitrogen content of the soil. Is it correct or not?

Khalil


Khalil,
Sure. You can specify any initial conditions for solute concentrations that you want. Thus if you have data from which you can identify the initial condition, certainly do so. The example that I sent you was a hypothetical example, in which I assumed that initially there was no solute in the transport domain, and all solute was delivered subsequently through fertigation.

Jirka


Hi, Jirka
Thanks for your reply. After specifying the value of cRoot in Reaction Parameters for solute I am getting good results. However, while giving the value of cRoot I could find four more columns as cBand1, cBand2, cBand3 and cBand4. I referred the manual and helh menu to get information for cBand1 cBand2, cBand3 and cBand4. But I could not understand about these parameters.
kindly tell me about the values to be given under cBand1 cBand2, cBand3 and cBand4 in case of surface drip fertigation.


Khalil
These values are used to specify concentrations for the time-independent (water flow) boundary conditions.
Jirka


**ID = 305, Axisymmetric flow in vertical direction**

I am a hydrus2D user. I have the following question to the forum; which are related to axisymmetric flow in vertical plane.

The dimensions of the slice of the aquifer are 100m (length) and 40m (deep). I have an atmospheric boundary on the top for the full length of 100m; which includes precipitation, evaporation, transpiration, solute transport (with non eqilibrium adsorption,biodegradation and production). I have to represent the flow to a well and pumping from the well; which will affect the water-table level

1. How can I represent a well?
valliyappan.t

Since you are using axisymetrical option, you need to well in the middle or the domain (on the left side). There you can use any type of BC appropriate for your application, i.e., specified head, flux, or seepage face.
J.

Choosing the well in the domain(left side):
I am choosing the well in the domain (left side) by selecting three nodes; which will be on the z-axis,then
1. what will be the well radius?
2. Does Hydrus2d assumes a zero well radius? (or) I have to choose a non-zero well radius?

valliyappan.t

You should choose a non-zero radius.
J.

Choosing a non-zero well radius
If i want to choose 1.5m radius, then how can i represent the well?
1. will the z-axis be the centre for the well diameter?

valliyappan.t

Sorry.
Had got the answer for the following question from the Hydrus2D manual.
Choosing a non-zero well radius
If i want to choose 1.5m radius, then how can i represent the well?
1. will the z-axis be the centre for the well diameter?

valliyappan.t

**ID = 306, Hydrus2D for Linux/Unix**

Hello,
Is a version of Hydrus2D available, which is compiled for Linux or Unix?
Lies

Hi Lies,
no, Hydrus runs only under Windows.
Mirek

**ID = 307, Axisymmetric vertical flow and well – Example**

I am a Hydrus2D user.

I am working on a model; which is an axisymmetric vertical flow towards a pumping well. I have problems with the simulation. I am working on this for a couple of weeks.

Is it possible for me to get a solved example problem (any reference paper, sample problems, etc related to Hydrus2d software) dealing with axisymmetric vertical flow towards a pumping well. It would be a great help for me to proceed further.

valliyappan.t


There are several examples distributed with the Hydrus2D that use axisymetrical option (although none is a flow to a well).

Direct/test7 - ponded infiltration
Inverse/disc - tension infiltration
Inverse/crust - tension infiltration
Inverse/cone - infiltration from cone penetrometer.
Check these project to get some guidance on how to proceed.

Jirka


## ID = 308, Problems with a large model domain

Dear Jirka
I am simulating nitrification in soil and groundwater from leaching of septic tank effluent from sock-holes, which are 4 m in depth. The diameter of sock-holes is about 1 m. I examine its cumulative impact in a community within 4 km distance.

I have split the problem into two models. The first model has a depth of 4 m with free drainage at lower BC. I use Hydrus-1D for this model.

The second model gets rid of soil layers above 4 m and thus the sock-holes are assigned as point sources at the top boundary with constant flux. The areas between sock-holes are variable flux BC using the fluxes at the bottom of the first model. I use Hydrus-2D for this model. Model domain is 4000 m in X direction and 100 m in Z direction. I first tried using variable density grids (sparser away from sock-holes and denser around sock-holes) but I always failed to run the model. At end I just use equal density grid for the whole domain and it runs although it is extremely slow.

I have a few problems that I wish to ask your advice.

Why I cannot insert observation points?
What is the reasonable value for dispersivity for such large model domain?
Should input of effluent flux be signed as negative rGWL values?
Is 10 m for hCritA value OK?

What are the options for reducing the running time? Is there any way to generate a better mesh for such problem? Are the time discretizations that I have used reasonable?

I have sent my input files for your examination. Is there any way to attach file in this website?

I am looking forward to hearing from you.
Thanks a lot for your advice.
Best wishes
Liping

Liping,
Your mesh looks strange. Did you really intend to have only one element covering the entire depth of 4 meters.
You can enter observation nodes. You need to have a node (from FE grid) inside of the cursor when you make your selection. It's working fine, I tested that on your example.
We typically recommend dispersivity on the order of 1/10 of the length of the transport domain.
Variable flux is positive for flow out of the domain.
10 m is very low for hCritA. But you need to realize that this value is used only when evaporation is simulated, which it is not in your case.

Jirka

quote:
--------------------------------------------------------------------------------
Originally posted by Jirka
We typically recommend dispersivity on the order of 1/10 of the length of the transport domain.
--------------------------------------------------------------------------------

Hello Jirka,
Can the ratio 1/10 of the dispersivity be also applied for paddy soil?
Thank alot,
lea

I do not have any specific information for paddy soils. However, if there is a lack of information, I would use this ratio.
Jirka

**ID = 309, Inner Boundary Conditions at Layer Interface**

Dear Hydrus users,
I am running a 1-d water flow for a domain consisting of 2 different soil types.

I want to apply a boundary condition at the inner interface between 2 soil layers, but Hydrus only allows me to define the boundary conditions along the boundary.
I tried to add a line at the interface in the geometry editor, but the line did not appear in the boundary condition editor.
I also tried to draw the domain boundary to include the interface, but Hydrus did not allow the overlapping points.
Is there a way to define the boundary condition at the inner interface?

Preecha


Preecha,
No, there is no way to have boundary conditions inside of the transport domain. Boundary conditions can be specified only on boundaries.

There are only two things which you can do to specify internal sinks or sources:

a) You can select nodes inside of the transport domain and specify a flux recharge to them. Such nodes will then act as sinks or sources (depending on the sign of the flux).

b) You can select nodes and fix pressure in them. This is, however, not supported by the GUI and you will need to do that directly in the Domain.dat file (convert the binary file domain.in into the ASCII file Domain.dat using Boundary module). Specify Kode for selected nodes equal to 0 and initial head to head you want.

Jirka


### ID = 310, Units

hi,
I'm a little bit confused with units.
- my atmospheric boundary is 3,6 m broad. as an example, I water the model on day 1 with 1000 mm rain. The result in Cum_Q.out is for CumQAP .36609E+7, so this is 1015 mm - looks ok.
- I create a lower free drainage boundary (very high ks for draining all the water after 1000 d). Independent of the with of the drainage boundary, the result in Cum_Q.out is always about ,36E+7 -> so that makes only sense when calculating ,36E+7 / 3600mm -> about 1000 mm, so output=input.

I looked into the hydrus book. I thougt, that for calculating the flow volume, I have to consider the length of the boundary - that is in my example the lenght of the free drainage boundary - but that is not 3600 mm ....

what is wrong?
regards - uwe


Table 11.3 from the manual should explain units for cumulative fluxes across boundaries. It is alway total volume (across the entire boundary). Only the actual

fluxes (which is in table 11.2) are for selected boundaries given along 1 lenght unit of BC (such as atmospheric BC).

Table 11.3. CUM_Q.OUT - total cumulative water fluxes.*

CumQAP Cumulative total potential surface flux across the atmospheric boundary (Kode(n)=±4), [L2] or [L3]+.
CumQRP Cumulative total potential transpiration rate, [L2] or [L3]+.
CumQA Cumulative total actual surface flux across the atmospheric boundary (Kode(n)=±4), [L2] or [L3]+.
CumQR Cumulative total actual transpiration rate, [L2] or [L3]+.
CumQ3 Cumulative total value of the bottom or other boundary flux across part of a boundary where the groundwater level, the bottom flux, or other time-dependent pressure head and/or flux is imposed (Kode(n)=±3), [L2] or [L3]+.
CumQ1 Cumulative total value of the flux across part of a boundary along which time-independent pressure heads and/or fluxes are imposed, including internal sinks/sources (Kode(n)=±1), [L2] or [L3]+.
CumQS Cumulative total value of the flux across a potential seepage faces (Kode(n)=±2), [L2] or [L3]+.
CumQ5 Cumulative total value of the flux across a boundary containing nodes for which Kode(n)=±5, [L2] or [L3]+.
. .
. .
. .
CumQN Cumulative total value of the flux across a boundary containing nodes for which Kode(n)=±NumKD, [L2] or [L3]+.


+ For plane and axisymmetric flow, respectively
* Boundary fluxes are positive when water is removed from the system.


**ID = 311, What does hydrus do about specific storage?**

Am I being blind or does Hydrus not require a value for specific storage? In which case how does it model saturated conditions?


No, Hydrus does not consider specific storage. Specific storage is always negligible compared to hydraulic capacity for the unsaturated zone. As a result, the solution for the saturated part is always instantaneous equilibrium.
Jirka


Thanks for that. Its a shame it doesn't do the saturated flow part properly as within the same model I'd like to model the unsaturated part as well as the propagation of pressures through saturated low permeability clays beneath.

I have just emailed you the Hydrus code that will allow you to enter specific storage coefficient once you start executing the program. I have it implemented in the code, although I still do consider specific storage to be irrelevant in the variably-saturated code.
Jirka


## ID = 312, Bulk density in Rosetta model

Hi
I'm using Hydrus 1D to modeling water movement in an unsatured profile. To obtain hydraulic parameters I'm using neural network prediction (Rosetta model); in particular, I'm using the correlation of hydraulic properties with %sand,%silt,%clay and bulk density.
My question is: the bulk density is the density of dry soil or the soil with natural water content?
Thanks


It is the density of dry soil.
J.


## ID = 313, Practical application

I am an Undegraduate in Sri Lanka. we are doing a project using HYDRUS 1D. we calibrate the soil hydraulic parameter( n, alpha )to inwestigat the,
1. variation of normalized ET with time for both saturated and
Unsaturated soil column
2. Formation of an extinction depth unde constant ET, and
3. Equilibrium soil profiles at differnt water depths

Then I have 2 questions please anyone help me
1. What are the real definitions of "n" and "alpha" ? (please
discribe)
2. What are the practical aplication of finding aboe 3 results

please help me my e mail " pereraprr@yahoo.com"
Rasika
## ID = 314, Constant head boundary - axisymmetric well problem

I have modelled an aquifer with 125m (length) and 40m (deep). I have a well which is on the left side of the domain. I have used a constant head and variable flux boundary to represent the well. A constant head boundary on the right side of the domain to represent the flow outside the domain. Atmospheric boundary on the top. The aquifer is subject to axisymmetric flow towards the well. The simulation is for a period of 1 year.

When I look into the pressure head graph on the post processing side, it gives me a straight horizontal line (say at 9.3m).

I would like to know what does this represent?
Is it at a point? which point?
Is it across the two head boundaries?

valliyappan.t


Hydrus will give you average pressure heads at different boundary types, i.e., one graph for constant in time boundary, one for time-variable boundary, one for seepage face, etc.

Jirka


## ID = 315, Initial condition

Hi,
I have a problem of initial condition for my simulation.
My experience is carried out by a satured sand column (l = 50 cm) with water table e = 2 cm.
How to simule the initial condition? Someone help me, please!

Adam.


## ID = 316, Infiltration concentration and variable head

Hello,
I appear to get a strange effect when I am simulating infiltration from a variable head boundary in the presence of an atmospheric boundary. For some reason the concentration of the infiltrating water is zero when no precipitation occurs, and the prescribed value when precipitation does occur. When I have a constant head boundary, the simulation is oke. For the moment, I solved the problem by replacing all zero precipitation values by very small dummy values. Is this a known problem, or do I have some error in my data?

cheers
lies
HYDRUS-2D allows you to enter three concentration values when using time-variable boundary conditions, i.e., cValue1, cValue2, and cValue3. The water flux across the boundary is calculated as q_top=Prec-Evap, and solute flux as the product of q_Top time cValue2 or cValue3. With cValue1 it is different. In that case I want to take into account possible increase of concentration of infiltrating water when evaporation is present at the same time as precipitation. In such case cValue1 is recalculated as follows:

if((Prec-Evap).gt.0.) then
cValue1=Prec/(Prec-Evap)*cValue1
else

cValue1=0.
end if

Thus if Evap is greater than Prec, there is no solute entering the profile, since there is no water entering the profile.

Jirka

## ID = 317, Model running time

Dear Jirka
I have successfully completed the calibration and validation of Hydrus-2D taking simulation period 48 hours and Minimum time step as 0,Maximum time step as 2 hour. After calibration and validation I want use this model for prediction of water and nitrogen distribution in the growing period of onion crop 127 days (3048 hour). In this case again I am taking Min and Max time step as 0 and 2 hours because irrigation is applied for 30 minute on alternate days and fertilizer is applied through drip system for 10 minute on the first day of each week. When I am running the model it did not complete the solution even after 12 hours. With this information I would like ask following:

1. Wheather the time step selected by me in this case is correct?
2. Wheather the time taking model for given solution is on expected line?
3. Kindly suggest me if there is any other way to manage this problem.

Thanking you
Khalil

The simulation time depends strongly on several factors, mainly:
a) the number of FE nodes
b) nonlinearity of the soil hydraulic properties
c) how smooth the boundary conditions are.

Thus withoug knowing this information, it is difficult to predict the simulation time.

I believe that for majority of simulations the following parameters provide the optimum solution:

tInit=0.001 d
dtMin= 1 s
dtMax = 0.25 d
MaxIter = 10
Water content tolerance = 0.001
Pressure head tolerance = 1 cm

Jirka

Hi Jirka

On your advise, I could solve the problem of model running time thank you very much for the same. I have one more query with the value of Croot. I am running the model for 127 days with drip fertigation in onion crop and I am applying fertilizer through drip at the beginnig of every week, in this case

1. whate value of Croot I sould take. Wheater it will be a constant value for whole growing season or will it change with time?

2. Dose the value of Croot chosen has any relationship with Root Water Uptake?

Khalil


cRoot is a constant for the duration of one Hydrus simulation.
It represents the maximum solute concentration that can be taken up by plant roots by passive uptake. As long as the actual concentration is lower than cRoot, roots take up water with all solute disolved in it. Once the actual concentration is larger than cRoot, only cRoot is taken up and the rest of solute stays behind in the soil.

Thus if you do not want to have solute uptake, set cRoot equal to 0. If you want roots to take up whatever solute is dissolved in water, set cRoot equal to a large number.

Jirka


## ID = 318, Kind of extraction

Dear
I would like to as about solute tranport in soil under unsaturated condition.
what kind of extraction ( pore water concentration or 1:1 soil water extract) I have to use to compare with Hydrus 2d findings.
thanks in advance.

yours Ahmed


## ID = 319, Flux at reference level

Hi,
I'm simulating water flow in unsaturated medium 12m width and 6 m height, I'm asking if is it possible to obtain water flux calculations (print out) at a reference level (e.g., z = 1m) ?

Thanks


Hydrus evaluates fluxes only across boundaries, but not over internal curves. There would be two way how to get to such number.

a) You can specify subregions, and then in the balance.out file, there is information about fluxes in/out of the subregion. If you know the flux at the surface, then from this and the change of storage in the subregion, you can evaluate flux at a particular depth.
b) Hydrus evaluates flux at each node. Thus in the output graphics you can make a crossection and then import fluxes along this crossection to some other software (e.g., excel) and integrate them along the crossection. Thus would, however, be less precise.

Jirka


Hi Jirka,
Thanks four your answer, the first method a) works well for calculating water flux at a reference level, I tried it for a simplified case of a 2D rectangular domain (X=3m, Z=2m) with a contant flux at the surface (Q0=0.05 m/day for X between 0 and 1.5m) and a free drainge condition at all the bottom boundary (X between 0 and 3m). The flux is null elsewhere. Using the water storage DV/DT(from balance.out file) and the flux at the surface I can simulate perfectly the drainage flux at the bottom (Qdrain=Q0-DV/DT), but my question what is the information that can I have from the Inflow values (in the balance.out file) to calculate drainge flux at bottom since these are very different from DV/DT?

Z. SAADI


Saadi,
The information "Inflow" in the balance.out file is basically information about the change in storage in the domain (subregion) over one time step, i.e., DV/Dt, and thus considers fluxes across all boundaries, including uptake.

I do have actually a version of the code (not supported by graphics), where one needs to specify nodes on the interface (plus elements on one side) and the code calculates flux (actual and cumulative) through this interface (line through the domain). Send me email if you are interested in this version (I'll be, however, traveling until next Tuesday, so do not expect answer soon).

Jirka

**ID = 320, Version differences**

Hello Jirka,
I am using version 2.05 of Hydrus-2D. What are the differences with the newer version 2.102? Is it useful to upgrade?
thanks Lies


Not big changes. Mainly some small bugs were fixed.
J.

**ID = 321, Problem in simulating with drain**

Hello,

I'm triing to find the stady state situation for a drain problem. The aim was to find the hight of the water level (where q = qs) and between two drains. How can I simulate this? I tried to put some precipitation (0.6), and have put a boundary condition on the drain (flux=-30). My units are cm, minutes and the distance between the two drain is 100 cm. Is it better to use Mesgen or not? I have tried with Mesgen, but the simulation is very long and often crashes. How can I improve the simulation in order to find the result? (I tried to change the points number in Meshgen, but without notable result)
Thank you a lot.


Hello
If I understand your request, you want to simulate water high (water table) in a drained soil. In this case, you have two choices: describe your soil profile and locate the tile pipe with Meshgen, put seepage face for tile pipe, and prescribed flux on top, or use drain option described in manual. They gave the same results (except near the drain).
In fact according Hooghoudt formula, Water Table is linked to the flux R by
R(precipitation)=Q(discharge)=K.H²/L² plus Keq.D.2H/L²
where K is saturated hydraulic conductivity, H water table, L drain spacing/2, Keq equivalent hydraulic conductivity, D equivalent depth if the drain depth is different from impervious layer. If drain lyes on impervious layer R=K.H²/L² and H=squarre(R.L²/K)
With this, you can forecast water table according precipation flux.
If you compare theory of Hooghoudt and HYDRUS, water table is very close.

Julien
PS Happy New Year to all HYDRUS Users !!!


**ID = 322, Surface Runoff**

Dear HYDRUS Users,
I have a question regarding how to obtain surface runoff from Hydrus with atmospheric b.c.
Since potential surface flux is the net flux between potential evaporation (PT) and precipitation, precipitation applied to surface is reduced by PT when precip > PT.
My question is whether surface runoff is calculated by subtracting the diference between the potential and the actual surface flux, or the difference between precipitation and the actual surface flux.

If runoff is the difference between potential and actual surface flux, it seems that some of precipitation was taken by evaporation when no evaporation is reported.

Kind regards,
Preecha

Preecha,
Hydrus indeed takes precipitation and evaporation values, subtracts them, and applies the result as the boundary flux. As such it evaluates actual and potential fluxes across the boundary, but not the actual and potential infiltration, evaporation, and runoff values. I have realized that these values (the actual and potential infiltration, evaporation, and runoff values) can potentially be of interest to many users, and started evaluating them within the program. In the latest versions (I believe), I do print them to the output, but do not draw them in the graphical interface. You should be able to find these values in the v_mean.out and CumQ.out files for Hydrus-2D and in the t_level.out file for Hydrus-1D (the columns at the end of the file).

Jirka

## ID = 323, Cyclic water flow and solute transport

Dear Prof. Jirka
I have a question regarding the cyclic simulation of water flow and solute transport. I simulated some problems, but the results are not consistent.
I ask about how can I simulate this problem especially the B.C.
Dr. Gamal Khalil

Gamal,
If you repeat boundary condition at a certain time interval, you should eventually reach a quasi steady state, when processes in the soil profile (heads, water contents, concentrations, outflow fluxes) will also repeat periodically. How long it takes to reach such a state will depend on the soil texture, i.e., will be relatively quick for sands and loams, but will take quite a while for clays.

Jirka

## ID = 324, Cyclic simulation - 100 years

I am a Hydrus 2D user (version 2.05). I would also like to add a few more specific questions in regard to cyclic simulation of water flow and solute transport which I came across in my parametric study of axisymmetric flow towards a well model.

The dimensions of the slice of the aquifer are 100m (length) and 40m (deep). The base of the aquifer and the external boundaries of the aquifer were modeled as no-flow boundaries. An atmospheric boundary above the aquifer allowed recharge due to rainfall, potential evaporation and the source for the tracer (1 mole/d). A constant head and seepage face boundaries were used to represent the well (along the left side

of the flow domain) in a time dependent saturated-unsaturated aquifer. The water table was initially set at a depth of 6m from the ground surface

Assumptions made in this modeling work are as follows: (1) No regional flow; (2) Isotropic Ks, (3) Zero potential transpiration and tracer uptake by plants, (4) Uniform adsorption coefficient for iron oxide-coated silt and sand and (5) Linear adsorption isotherm.

The simulation was carried out for two types of systems (homogeneous and heterogeneous) for a period of 100 years (cyclic process in which 1 year data is cycled 100 times).

The different types of models simulated for two cases (1) no adsorption and (2) with Kd = 0.1 m3/kg are as follows:
1. Homogenous system – sand (Ks = 43.2 m/d)
2. Homogenous system – sand (Ks = 7.128 m/d)
3. Homogenous system – sand (Ks = 0.864 m/d)
4. Homogeneous system – silt (Ks = 0.0864 m/d)
5. Homogeneous system – silt (Ks = 0.06 m/d)
6. Heterogeneous system - both sand and silt (first 10m silt; rest 30m sand)
7. Heterogeneous system - both sand and silt (first 20m silt; rest 20m sand)
8. Heterogeneous system - both sand and silt (first 30m silt; rest 10m sand)

In the parametric study, all parameters were kept constant (which means the saturated water content, residual water content, I, n and alpha were not changed for sand and silt) except Ks as mentioned above for VG-M model and Kd for solute transport.

My questions to the forum are follows:

1. Atmospheric boundary head plot (time in days vs head in meters)

For sand: I am getting reasonable results (no changes in the result, the 1 year results are repeated 100 times; the results are cyclic)

For silt: I am NOT getting reasonable results (very high changes in the first 15 years and then the results are cyclic for rest of the 100 years)

** WHAT AM I DOING WRONG? WHY IS THIS HAPPENING? ANY SUGGESTIONS WOULD BE GREAT.


2. Plot of time in days vs concentration in microgram/litre

The values were taken from the observation nodes placed on the well screen at 4.5m deep, from 5 homogeneous model with no adsorption and 5 homogeneous model with adsorption (Kd = 0.1 m3/kg).

The results are very strange. The concentrations in the well for different Ks values are as follows:

Ks=0.06 m/d > Ks=0.0864 m/d > Ks=7.128 m/d > Ks=0.864 m/d > Ks=43.2 m/d

But in reality it should be

Ks=43.2 m/d > Ks=7.128 m/d > Ks=0.864 m/d > Ks=0.0864 m/d > Ks=0.06 m/d

** WHY IS THIS HAPPENING? ANY SUGGESTIONS WOULD BE GREAT

3. Plot of time in days vs concentration in microgram/litre

The values were taken from the observation nodes placed on the well screen at 4.5m deep from 3 heterogeneous model with no adsorption and 3 heterogeneous model with adsorption (Kd = 0.1 m3/kg)

The results are very strange. The concentrations in the well for different Ks values are as follows:

Sand: Silt (1:3) > Sand: Silt (3:1) > Sand: Silt (2:2)

But in reality it should be

Sand: Silt (3:1) > Sand: Silt (2:2) > Sand: Silt (1:3)

** WHY IS THIS HAPPENING? ANY SUGGESTIONS WOULD BE GREAT

I am still working on this topic. Expecting some suggestions which am hoping will put me in right direction.

Thanks for your time and help.

thank you once again
valliyappan.t

Always check the mass balance in the balance.out and Peclet numbers in run_inf.out. If mass balance errors are bigger than 2% or peclet numbers are larger than 5, then you need to use finer discretization. If mass balance errors are small and peclet numbers are below 5, then you most probably are getting the right answer to your problem.

Jirka

**ID = 325, Surface Runoff from HYDRUS2D**

Hi All,

I would like to simulate surface runoff (in addition to other outputs) using HYDRUS2D. I do not know how I can turn on the program so that I am able to get the runoff output. In the v_Mean.out file I only see the column for runoff with zeros. Does anybody know how I can get the runoff from the model?
Thanks,
Adion Chinkuyu.


Adion,
If runoff in v_mean.out is reported as being zero, then there is no runoff predicted during the simulation.
How to get runoff?
a) You could decrease conductivity of the top soil
b) SPecify precipitation fluxes of larger intensities (shorter time intervals).
Jirka


Jirka,
Thanks for the response. I am now getting some runoff.
Adion.


## ID = 326, Heat transport in 2-D?

Hasife, 01/31/2005
Can Hydrus2D simulate 2D heat transport?
Vertical and horizontal heat flow can be modeled?
Please, let me informed of it.


Rvang, 01/31/2005
Yes, it can simulate 2D heat flow. Perhaps have a look at the manual (download from this site) for a description of the processes. Thanks.--Rien van G.


## ID = 327, Diffusivity related concentration balance error

jkrivic 02/02/2005
Hi!
I have created a simulation of water flow and solute transport through 10 m thick layer of sand.
I am using Atmospheric BC with surface Run Off for the upper boundary and Constant Pressure Head with a value of 2 m for the lower boundary, thus assigning a water table level at the depth of 8 m. For the upper boundary I am using variable boundary conditions with different precipitation for each month, cyclically for a number of years. I get an infiltration dependant, cyclical water outflow from the lower boundary as expected. The water balance R is small (approx. 0.1%).

However, with the solute transport I get some very strange results.

I am injecting a constant solute concentration cTop for the first month of the simulation.

The Soil Specific Solute transport parameters are as follows:

Bulk.D.=2, Disp.= 0, Fract.=1, Thlmob.=0

I tried to assess the influence of diffusivity of solute on solute transport. I ran the simulation with several different values of the molecular diffusion coefficient in free water, ranging from 0 to 1000 cm2/day. For the values from 100 to 1000 I get reasonable results with concentration balance R of approx. 0,3%. But as the diffusivity parameter decreases, the CncBalR increases exponentially. When DifW is set to 0 the CncBalR is 60% and the mass outflow of the solute is 3 times higher than the initial inflow. With the DifW of 1 the CncBalR is 30% and with the DifW of 10 the CncBalR is 5%.

Is it necessary to assign a certain minimal value of DifW to keep the CncBalR low? Or am I doing something else terribly wrong?

Regards,
Jure


Jirka 02/02/2005
Jure,
Hydrus solves the convection-dispersion equation. It does not solve the purely convection equation. Consequently, you can not specify both diffusion coefficient and dispersivity as being zero. The code then encounters serious numerical oscillations. These oscillations can be encountered also when dispersive flux is much smaller than the convective flux. They can be avoided using some of the stabilization options, such as upstream weighting, stability index, etc. See Section 6.4.6. Oscillatory Behavior of the manual for more details.

Jirka


Jkrivic, 02/03/2005
Jirka,
Thank you for explaining that. Is it safe to assume that a certain marginal values of diffusion coefficient and dispersivity exist below which error starts to build up? How can I define those minimal values?

Jure


Jirka, 02/03/2005
The minimum values that the code can handle depends on the spatial grid. The finer the grid, the lower values of diffusion or dispersion the code can handle without introducing numerical dispersion. Look at the Peclet numbers in the time information. The code should be stable up to Peclet equal to 2 and acceptable results should be obtain for Peclet numbers up to 5.

Jirka

**ID = 328, Concentration unit**

Azhdary, 02/02/2005
Dear Jirka
Your variable suggestions helped me a lot. I am nearing to omplete my study. At this stage, I experienced one more problem this is regarding the unit of solute concentration in boundary condition editor. In help file the unit of concentration is given as (M/L3).I would like to know what is the unit of volume here? Whether it is Mliter or CC of water or soil volume. Further to add this I would like to ask you in which unit I should give initial concentration of nitrogen? That is in mg/ml of water or mg/cc of soil volume. Mass unit is mg.

Best regards
Khalil


Valliyappant, 02/02/2005
The unit of volume is cubic centimeter if you choose the 'Length' units in 'cm'. The results in the post-processing section will be in M/L3

I believe that, it is the concentration of solute in water for your problem. It can be soil volume if the solute is being desorbed (leaching experiment).

You can check the Topic 'UNIT CONSISTENCY'. The topic ID is :256. You can also see few other FAQ's 232,210 and 29.
YOU CAN ALSO WAIT FOR JIRKA'S REPLY. I am also interested to know whether the soil volume concept is correct.

Thanks
Mani


Rvang, 02/02/2005
Solute concentrations are in mass per unit volume of the liquid phase. --Rien van G.


Jirka, 02/02/2005
Concentration units in Hydrus are M of solute per volume (L3) of water [M/L3], i.e., concentration in water. In general, concentration units should be consistent with the length units [L] used in the rest of the project. Then the derived variables are as follows for 1D, 2D, and 3D problems, respectively:

Amount of solute in the soil domain (c*A): [M/L2], [M/L], [M]
Solute flux (qc): [M/L2/T]
Cumulative solute flux (qct): [M/L2/T]
Cumulative solute flux over boundary of length L (qctL): [M/L/T]
and so on. M can be anything, i.e., mg, g, mol, meq, etc.

If you, however, look at the governing convection-dispersion equation, you can see that the concentration term appears in each term. Consequently, the concentration [M/L3] does not necessarily have to be given in the same length units [L], as the transport domain. However, one then needs to convert variables derived from concentration (given above) considering this mismatch of units (between concentration and length units).

Jirka

Azhdary, 02/06/2005
Hello Jirka
Thanks for your replies. While finalyzing result, out of curiosity I executed the model by selecting the vertical plane and Axisymmetrical vertical flow separately.I got much different result in both the cases.I am simulating water and solute distribution under surface drip fertigation. Emitter to emitter spacing in the same line is 50 cm and line to line spacing is 60 cm. I want to simulate the water and solute distribution under single emitter. Kindly suggest me what type of flow I should choose from the list of the Horizontal plane, Vertical plane, Axisymmetrical vertical flow.

Thanking you
Khalil

Valliyappant, 02/06/2005
With same input parameters, same flow and transport conditions and same domain, you will get and you should get different results by just changing the type of flow in geometry dialog box.

This is because of the following reasons:

1. Horizontal plane is the areal view of your domain (LxL) - Both Z and X axis are Horizontal co-ordinates.

2. Vertical plane is the side view of your domain (LxL) - vertical co-ordinate is Z axis and Horizontal co-ordinate is X axis.

3. Axisymmetric vertical flow (LxLxL)- It is the symmetry of your domain along the vertical co-ordinate (rotating your domain along the y-axis; according to Hydrus2d it is z-axis). The left handside of your domain should coincide with or should be to the right of your vertical axis. In axisymmetric vertical flow, z is vertical co-ordinate and x is radial co-ordinate.

Hope this helps.
valliyappan.t

Jirka 02/07/2005

Whether to use 2D or 3D description really depends on the geometry. If drippers are on one dripper line, would use a 2D description (Skaggs et al. 2004). If drippers are far appart (such as for tree irrigation) then I would use axisymmetrical 3D description.

Look at the following manuscript that all deal with subsurface irrigation:

Skaggs, T. H., T. J. Trout, J. Simunek, and P. J. Shouse, Comparison of HYDRUS-2D simulations of drip irrigation with experimental observations, J. of Irrigation and Drainage Engineering, 130(4), 304-310, 2004.

Lazarovic, N., and J. Simunek, U. Shani, System dependent boundary condition for water flow from subsurfac source. Soil Sci. Soc. Am. J., 69(1), 46-50, 2005.

Gärdenäs, A., Hopmans, J. W., B. R. Hanson, and J. Simunek, Nitrate leaching under micro-irrigation for different fertigation strategies, Agric. Water Management, (submitted July, 2004, in press).

Azhdary 02/23/2005
Jirka
Thanks for your replies. Based on your advised, I did the simulation with Axisymmetrical vertical flow. we are using surface drip system. Emitter to emitter spacing is 50 cm and lateral to lateral spacing is 60 cm, emitter discharge is 3 liter/hr and crop is onion. With Axisymmetrical vertical flow we are getting better results. Out of curiosity I did the simulation with vertical plane also but results in this case are not enywhere close to observed valus. Kindly advise me on this issue.

Best regards
Khalil

Northbrightstar 02/23/2005
nice greeting from Malaysia
regarding to point source trickle irrigation how did you install the boundary condition of constant flux? if I choose the dimension of the emitter it would be overflow, then the description would be under head rather than constant flow, is it possible to take the saturated radius as a boundary of constant flux? and if the saturetd zone would not appear how can I select the boundary of constant flux?
thanks in advance

Jirka, 02/26/2005
If you have a surface drip irrigation then the surface boundary is dynamic, i.e., the saturated radius increases with time. The currently available version of HYDRUS-2D does not calculate this dynamic BC. You need to specify as boundary condition the final saturated radius and specify the constant flux on it. The radius of the saturated region you can estimat for example using Woodings analytical solution.

Jirka

**ID = 329, Well screen – bounary representation**

Valliyappant, 02/02/2005
I am a Hydrus 2D user (version 2.05). I have a question to the Hydrus2D forum. I would like to know the best way to represent the well (water abstracted due to pumping) screen boundary.

The dimensions of the slice of the aquifer are 100m (length) and 40m (deep). The water table was initially set at a depth of 6m from the ground surface. The base of the aquifer and the external boundaries of the aquifer were modeled as no-flow boundaries. An atmospheric boundary above the aquifer allowed recharge due to rainfall, potential evaporation and the source for the tracer (1 mole/m3/d).

A constant head and seepage face boundaries were used to represent the well (along the left side of the flow domain) in a time dependent saturated-unsaturated aquifer.

But I would like to use constant flux boundary instead of pressure head boundary to represent one-part of the well screen . Because i know the pumping rate and the amount of water being drawn. If i use constant flux boundary I can see water being pumped into the aquifer from the well during initial period of simulation, later the water is being pumped out from the aquifer.

I do not want the water to be pumped into the aquifer from the well.
How can I rectify this? Any suggestions would be great.
Thanks for your time and help.
mani

Jirka 02/02/2005 :
If you have the problem correctly defined, i.e., you use right special and temporal discretization, then you should get for the constant flux BC a constant flux either in or out of the domain during the entire simulation time, depending on the sigh of the specified flux. Positive flux goes in, while negative flux goes out of the domain.

Valliyappant, 05/20/2005 :
I have a question to the Hydrus2D forum.
The dimensions of the slice of the aquifer are 100m (length) and 40m (deep). The water table was initially set at a depth of 6m from the ground surface. The base of the aquifer and the external boundaries of the aquifer were modeled as no-flow boundaries. An atmospheric boundary above the aquifer allowed recharge due to rainfall, potential evaporation and the source for the tracer (1 mole/m3/d).

A constant head and seepage face boundaries were used to represent the well (along the left side of the flow domain) in a time dependent saturated-unsaturated aquifer.

The FEM models are used to give a resonable idea on water flow and contaminant transport for a given domain; which will usually represent an area in reality as a whole or part. So when a portion of the area of interest is taken for simulation I cannot use no flow boundaries on the sides because there is posibility of flow through sides as well. And the horizontal conductivity is usually larger than vertical conductivity (10:1).

I would like to know the best boundary condition to represent the regional flow for a given geometry.

It would be great if i could get some explanation for the following: Under what circumstances the constant head/flux and variable head/flux BC's are more suitable for practical applicability? One practical applicability example for each would be great (in words; not a simulation).

THANKS FOR YOUR TIME AND HELP.

Mr V.Thinnappan


### ID = 330, CBnd1 - con. of time-independent BC

Liping 02/02/2005
Sub: CBnd1 - con. of time-independent BC
Dear All
I have a model with a few time-independent boundary conditions – top with a constant flux BC (which is the contaminant source) in the unsaturated zone, and left and right with constant head BCs (not the contaminant source) in the saturated zone .

I assigned CBnd1=38 mg/l for the constant flux BC, and CBnd2=CBnd3 and the rest as zero concentration.

But I got high solute concentrations not only in the domain beneath the constant flux BC but also in the domain between constant head BCs. The unsaturated zone above the water table has very low concentrations.

It seems like CBnd1=38 mg/l was for all time-independent boundary conditions.

When I used non-flow left and right BCs, this problem can be avoided. Is this the only way to avoid the problem? But I think it should be constant head BCs for groundwater domain.

I hope to hear your advice. Thanks a lot for your help.
Liping


Jirka 02/02/2005
Liping,

As a default, everywhere on the constant head and flux boundaries, cBND1 is applied. But this can be changed in the boundary module. When you select the boundary condition (either Cauchy or Dirichlet), the code will ask you for a pointer to the vector of BC. Number 1 goes to cBnd1, 2 goes to cBnd2, etc.

Jirka


## ID = 331, Hydrus-1D tutorial – inverse problem

Baharper, 02/03/2005
I am unable to get output for the inverse problem in the Hydrus-1D tutorial online. I do not get an error message; however, it does not appear that the program is performing any interations when it is executed. I am following the input instructions and have two questions regarding them. For observation points, the instructions just read "Observation Points?"; so I inserted what I thought was an approriate number of observation points across the profile. Also, for the initial condition of pressure head, I selected the entire profile and entered a top value of -2 and a bottom value of 2.52 and deselected "use top value for both". However, I don't understand the next line, "Lowest node = -1000 cm". There is not an option for it in the dialogue box after selecting the entire profile. Am I supposed to select the "edit condition" button again and select the lowest node, which I have already given a value of 2.52? Everything else from the input instructions has been entered correctly. Thank you for your help.

Brian Harper


Jirka, 02/03/2005
Brian,
The tutorial is certainly correct. We have used it in tens of the short courses that we have given around the world.
Observation nodes: You can enter some. It is relevant for the run. It is just to show some results.
The bottom node must have initial pressure specified equal to -1000. That's how the outflow gets generated.

Jirka


## ID = 332, Flux

mizo316, 02/04/2005
Hi Jirka --
I have multiple injection pipes in a vertical cross-section domain. The pipes are equally distant on a single row. When I applied a constant flux of a given value "a" to all pipes, the total flux and the seepage flux for that simulation should be # of pipes times "a", would that be correct? I am getting much smaller value than the total imput flux i am assigning.

Thanks,

Mazen

Jirka 02/04/2005
Mazen,
The constant flux is applied over certanain lenght of the boundary. Therefore the total flux is "Constant Flux" * "Number of Pipes" * Circomference of one pipe".

Jirka

mizo316 02/04/2005
Correct Jirka, that is how i check it usually. However the mistake was in the input file. I got it fixed.

Thanks again,
Mazen

**ID = 333, Pressure head interval (ha, hb)**

valliyappant 02/05/2005
I have a question to the forum regarding the pressure head intervals (ha,hb)in the iteration criteria dialog box.

How do I choose the values for ha and hb?

I do know the range given in the manual. But I would like to know how it works in reality. OR the reason for choosing a certain value within the range mentioned in the hydrus2d manual.

It would be great if i can get some reference in regard to this.
Thanks for your time and help.
Thank you once again.
valliyappan.t

**ID = 334, How to choose a hydraulic model**

Sales, 02/07/2005
Hello,
Hydrus gives 3 possibilities for a hydraulic model: Van Genuchten-Mualem (with or without -2cm air-entry), Van Genuchten modified, and Brooks-Corey. I wonder in what situation one model is more appropriate than another. I know that the most

common model is Van Genuchten, and also to what formula each of the models refer. But I don't know in which case to use what model.

Thank you for your time.


Rvang, 02/07/2005

It all depends on how well the curves fit your data (assuming you have any retention/conductivity data). Of course I am somewhat biased by suggesting that the VG equations generally give slightly better results than BC for relatively medium- and fine-textured soils and, more generally, for soils with relatively broad pore-size distributions. For coarse-textured soils it does not matter much. For very fine-textured (clay) soils with n values less that about 1.2, to -2 cm option is important since it will avoid very nonlinear conducvity curves. Some of these things are discussed in the RETC manual (VG versus BC) and in a paper by Vogel et al (Adv. Water Resour. Res. 24(2): 133 144, 2001) about the -2 cm. Thanks.--Rien van G.


**ID = 335, Mesh generation**

Valliyappant, 02/08/2005
This question is based on the topic 'problem using hydrus2d' Topic ID 285.

If I have dupicative nodes in the corners,will Hydrus give me an immediate warning (for example like Geometry inconsistent) when i leave the geometry and FEM mesh editor dialod box?
OR
will it be a common error messsage before simulation (for example like invalid number or zero divide or floating point error)?

Thanks for your time and help.
thanks you once again.
valliyappan.t



Jirka, 02/08/2005
The older version of Hydrus-2D did not check for this problem and would give you the latter response.
The latest versions (since 1-1-04) should send a message: '
Error in FE mesh! Check that you do not have individual - nodes on the boundaries !'

Jirka

**ID = 336, 2 questions about capillary barrier and using BC**

Sales, 02/09/2005
Hello,
I have two questions:
I wanted to simulate a fictive two dimensional capillary barrier problem. For this, I took the example given in the Manual Projects for Hydrus, and I changed the

geometry: I used Meshgen to create a profile that resembles the one that can be found in Mesh-2. The simulation works ok, but there is only a vertical flux! But in a capillary barrier, you should have a water flux between the two soils (if you have some slope). So, I am wondering what I did wrong.

The second question is about Brooks and Corey: when I take the same soil (loam for example) and simulate it for the same amount of time, is it normal that the water content and charge versus time do not correspond to the other models? I am having some problem explaining these differences. I know that the parameters of the soil are different (like shown in the appendix 1).

Thank you very much.


Jirka 02/09/2005
The first part of your question:
I can say based on information you gave. Check the mass balance. If it is OK then what you got is most probably correct and the contrast in material properties is not sufficient to generate horinztal flow.
The second part: Brooks_Corey and van Genuchten functions do have different properties. For example air-entry value in the Brooks and Correy function. As a result, the simulation results may be quite different depending on the saturation level in your problem. Also compare the soil hydraulic functions (both retention and conductivities) and see how similar they are. Hydrus does not provide any quidence with respect to parameters for Brooks and Corey and thus look closely mainly on this.

Jirka


Sales, 02/10/2005
Thank you very much for your answer:
for the first question, the mass balance is ok , but the properties of the soil are very different: I have chosen the parameters proposed by Hydrus for sand and clay. But there is still no horizontal flux. What could still be wrong?

For the second question: I have compared the retention and conductivity function and they are different. Does that mean that the parameters proposed by Hydrus for a soil with BC are not necessarily correct?


Jirka, 02/10/2005
The soil hydraulic properties provided in the soil catalog of Hydrus are only as good as the source from which it was taken, i.e., the Carsel and Parich paper and the RETC program. I do not know how familiar you are with the physics of the capillary barriers. Obviously they do function as capillary barriers only for certain pressure heads. That means that if the pressure heads in the system are larger than certain value, the flow will be predominantly vertical, and if pressure heads are smaller than

this value, the flow will start divert horizontally. For clays and sands this value is about -30 cm. Do you have pressure heads smaller than this?

Jirka

Sales,  02/11/2005
You are right, I'm not really familiar with the capillary barrier. I have simulated a flux (near Ks of the clay),witch provoques a pressure head at the end of simulation of about -2 cm at the top and -15 on the bottom. My initial conditions were a charge of -100 every were. Does that mean that I should reduce the flux? (I didn't understand you when you talked about values smaller than -30: does it mean that I should have values like -40 or like -20). The only thing I wanted to obtain was a distance where the water is not going in the sandy soil. That means that the water should go no longer vertically for a certain distance. So, what is wrong?
Thank you very much

Jirka, 02/11/2005
Yo need to look at the conductivity functions of the two materials. The capillary barrier effect is due to the conductivity functions crossing each other at a certain pressure heads. Sands are more conductive in the water content range close to saturation than clays, however this reverses for lower pressure heads. Therefore, capillary barrier effect is not at all existant close to saturation, but becomes very important at lower pressere heads (more negative).

J.

## ID = 337, Mass balance error

Valliyappant, 02/10/2005
The mass balance error is one of the approximate method to check the accuracy of the numerical results.
After obtaining results from a number of simulations, I can see the following trends in those simulations

1. The mass balance error is below 1% at all print steps
2. The mass balance error decreases linearly from a high value (for example 10%) to a low value (for example less than 1%)
3. The mass balance error increases during the initial print steps and reaches a high value (for example 10%) then decreases from a high value to a low value (for example less than 1%).

I have two questions to the forum regarding the mass balance error. They are as follows
1. Can I consider all the above three trends correct?
2. What could be the reason for high mass balance error only at certain print steps?

NOTE: I HAVE MADE CHANGES TO THE TOPIC 'PRESSURE HEAD INTERVALS (ha,hb)'. IT WOULD BE GREAT IF I COULD GET SOME SUGGESTIONS IN REGARD TO THE SAME.

Thanks for your time and help.
thank you once again.
valliyappan.t


Ena, 12/20/2005
Hi
I was analysing the infiltration of rainfall into a 6 m high slope with an angle of 26 degrees. The geometry of the slope includes the crest, the slope surface and the toe. Initally, the water table was at -1m below the toe of the slope (with no inclination of the ground water level)

Slope was homogeneous and the default parameters of Loamy sand as suggested by hydrus was used. Rainfall was applied ontop of this slope at 0.044 m/hr for 25 hours.

For the first 6 hours, the mass balance error was < 1%. I noticed that as the water table at the base of the slope reached the ground at $t = 7$ hours, the mass balance error started to increase slowly until it reached 39% at $t = 25$ hours.

I changed the density near the surface but it doesn't solve the problem. I also tried to minimise the amount of rainfall (0.3 mm/hr) and run it for a longer hours, and once the water table at the base of the slope reached the ground surface, the mass balance started to increase again. Please could anyone advise me in what other aspects should I look at in order to prevent this from happening?

Many thanks
Ena


**ID = 338, Water balance**

Paoloc, 02/11/2005
I am having some problems with the water balance, so I was wandering about this possibility:

Let us imagine a scenario with some potential evaporatin defined by the user. If ponding builds up at some point, will water evaporate from the pond (at a rate defined by the potential evaporation)?

thanks
paolo
Jirka, 02/11/2005
Paolo,
If you use "the atmospheric boundary condition with the surface layer" then yes, during ponding when water layer exist at the soil surface, the water layer is reduced due to both infiltration into the soil and potential evaporation.

Jirka


**ID = 339, Observation nodes – insert**

Valliyappant, 02/12/2005
The dimensions of the slice of the aquifer are 100m (length) and 40m (deep).

I used general option in the geometry dialog box to create the geometry and the mesh. I am not able to insert observation nodes at certain parts of the domain, particularly where i have increased the node density.

I can insert nodes in rest of the parts of the domain.
How can I rectify the problem?
Thanks for your time and help.
valliyappan.t


Mirek, 02/12/2005
Hi, I have two questions:
1/ Please describe more precisely where is the problem (what you are doing and how the program responds)
2/ What is the FE-mesh cell size close to the point where you want to have the observation point?

Mirek


Valliyappant, 02/13/2005
The dimensions of the slice of the aquifer are 100m (length) and 40m (deep). I have axisymmetric flow towards well. The well is placed on the left handside of the domain. The well is 0.2 m diameter and 15 m deep.

I need to insert observation point on the well screen at 4.5 m, 9 m and 15 m deep. The density at the nodes which make up the well are 0.3 to the left and right.

My mesh was very fine at the points where i need to insert observation nodes. So I zoomed the domain 3 times I suppose. Then I tried to insert nodes at these points. When I clicked at 4.5 m deep the observation points where not inserted at 4.5 m but was inserted at may be 20 or 25 m deep.

The other problem I encountered was the X and Y co-ordinate values on the right corner of the window didn't work when I was having the zoomed image. But it did work in the normal view.

I think the FE-mesh cell size was about 2 to 5 mm maybe.

HOW CAN I EXACTLY MEASURE THE FE-MESH CELL SIZE FOR REGULAR AND IRREGULAR GEOMETRIES? WHAT IS THE USUAL METHOD FOR THIS?

NOTE: THE VERSION I AM USING IS HYDRUS2D 2.05, 1999. I DID TRY IN A DIFFERENT VERSION (NOT VERY SURE ABOUT THE VERSION NUMBER) RELEASED IN 2001 I SUPPOSE. IT WORKED PROPERLY.

Thanks for your time and help.
thanks you once again
valliyappan.t


Jirka, 02/13/2005
You probably have an option in "Grid settings" set on "Snap to grid" (in the Boundary Module). Deselect that, and then you should be able to specify observation nodes in any node of teh finite element grid.

Jirka


Mirek, 02/13/2005
Hi Valliyappan:
Please check what Jirka suggested. If this doesn't solve your problem then I think that your mesh is simply "too fine". If X and Y coordinates on status bar don't change when you are moving mouse pointer over the zoomed view (for example from point A to point B) then it means that the space difference between A and B is less than 0.001 m (if your geometry is defined in meters). At the same time dimensions of your whole domain are 100x40 meters. It means that the ratio $Rm=Dmax/Dmin>1.e+5$ (where Dmax is diameter of the whole domain and Dmin is diameter of your smallest FE-mesh cell) is too large (I'd recommend $Rm<1.e+4$). Although all calculations are performed in double precision I think that there can be problems with rounding errors. Now there are probably some problems while inserting observation points but I'm almost sure that later there would be other problems (during calculation, etc.). Also your mesh has probably a very large number of elements (a reasonable number is $Ne<20000$) and therefore calculation would be very time-consuming.

I'd recommend you to modify FE-mesh parameters so that the final mesh had:
1/ $Ne<20000$
2/ $Rm<1e+4$

Regards Mirek


valliyappant 04/19/2005
Dear Jirka,
I would like to measure the porewater presssure at more than 10 points/places within my flow domain. But the Hydrus2d version 2.0 which am using allows me to have only 10 observation points at the maximum.

I read the descriptions of other latest versions in the hydrus2d website. I couldn't see any description about the increase in number of observation points in the above versions.

Is there any other option by which I can measure the porewater presssure at more than 10 points/places within my flow domain.

Any suggestions would be great.

Jirka 04/19/2005
First, I believe that our latest verstion that can be downloaded here allows up to 30 observation nodes.
You can also try to do that manually in the older versions.
Increase the number NObsD in the dimensio.in file, NObs in the Boundary.in and specify nodal numbers for observation nodes.
Most probably this will work too.
Jirka

valliyappant 09/28/2005
I want to transfer pore water pressure data from hydrus2d to slope stability software.
It would be helpful if i know the x and z coordinates for the nodes in the domain.
How can i obtain the X and Z coordinate for the nodes in the domain?
Thanks for your time and help.
Mr V.Thinnappan

Mirek 09/28/2005
You can export FE-Mesh to an ASCII file (the Meshgen module) and then you can read nodal coordinates from the file.

Regards
Mirek

**ID = 340, I type BC**

Rajandrea Sethi, 02/14/2005
Hi!
I have some problems with CXTFIT when using first type BC with "Constant initial concentration".

It always gives me this error:
WARNING ! Z0 > 0 FOR A FIRST TYPE INLET
I've avoided this problem using a dummy stepwise initial distribution, but wondering if is this a bug or I'm somewhere wrong...
Is there anyone that could help me?

Ciao!
Raja


Jirka, 02/14/2005
Raja,
Frankly, It is not clear to me why you have posted this question here in the discussion forum today. You have contacted both me and Rien personally last week and Nobuo Toride, the developer of CXTFIT2 program, solved this problem and send you the solution two days ago.

Jirka


## ID = 341, BC switching and more

AlexFurman, 02/15/2005
Hi all,
Any ideas on how the following can (if at all) be done:
1. have two different sections of a boundary have the same type of conditions (say variable flux) but with different values (for the water flow part)
2. Switch during simulation between a variable flux BC and no flow BC, or head BC (other than manipolating the binary files down there)

Thanks,
Alex


Valliyappant, 02/15/2005
Hi Alex,
From what I understood from your question I am giving the reply.
Reply for question (1)

NOTE: FOR VARIABLE FLUX BC YOU WILL NOT BE ABLE TO SET VALUE. DEPENDING ON THE INITIAL CONDITIONS SET FOR THE DOMAIN THE VARIABLE FLUX IS CALCULATED. IT IS A TIME DEPENDENT BC. YOU CAN SET VALUE FOR CONSTANT FLUX BC.

First select the constant flux BC option from the left hand side of the window. Select the first section of the boundary (Selection can be made by clicking on the initial node of the section and finishing by clicking on the last node of the section). Once the

selection is made you will get a dialog box where you can give the value for the flux. Select the second section and do the same.

Reply for question (2)
Since variable flux is calculated based on the initial conditions of the domain and is time dependent, it can act as flux and no flow boundary based on the internal flux in the domain.

Hope this helps.
valliyappan.t


Jirka 02/15/2005
Alex,
1) If you need to specify two time-variable flux BCs, then use atmospheric for the first one and variable flux for the second one.
2) Hydrus in general release can obviously switch boundary conditions from variable flux to zero flux (you set flux equal to zero), but it can not switch from variable flux to variable head (except for atmospheric BC due to ponding and excessive evaporation).

Jirka


AlexFurman 02/15/2005
Thanks valliyappan and Jirka - I guess I was not clear enough.
The idea is that two different sections of the boundary have variable boundary conditions. Idealy both will have atmospheric BC that switches from time to time between head and flux, but as much as I know this is not possible, so was wondering of peoples experience with the alternatives, as valliyappan and Jirka suggested

Thanks
Alex


## ID = 342, Simulation time

valliyappant  02/15/2005
The dimensions of the slice of the aquifer are 100m (length) and 40m (deep). I have axisymmetric flow towards well. The well is placed on the left handside of the domain. The well is 0.2 m diameter and 15 m deep.

The water table was initially set at a depth of 6m from the ground surface. The base of the aquifer and the external boundaries of the aquifer were modeled as no-flow boundaries. An atmospheric boundary above the aquifer allowed recharge due to rainfall, potential evaporation and the source for the tracer (1 mole/m3/d).

A constant flux and seepage face boundaries were used to represent the well (along the left side of the flow domain) in a time dependent saturated-unsaturated aquifer.

PLEASE NOTE THE FOLLOWING:
1. The initial and minimum time step are 0.0001 of a day.
2. The number of elements in my domain are 12125.
3. The simulation is run for a period of 100 years.
4. I have 1200 time dependent input data's for atmospheric BC in which first 12 input data's for twelve months of the first year are same for the rest of the 99 years.

I am concerned about the simulation time. It took me one full day (day and night) to run the simulation for 3 years time period. The simulation is still running. It has to run for another 97 years time period. The computer I am using is P4.

HOW CAN I REDUCE THE SIMULATION TIME?
IS THERE ANY OTHER OPTION TO REDUCE ATMOSPHERIC BC INPUT DATA?

Any suggestions would be great.
thanks for your time and help.
valliyappan.t


Jirka 02/15/2005
Optimal time step for these highly nonlinear systems depends on two many things, mainly the spatial discretization. I have recommended somewhere on this site the values for other parameters that I would use to acchieve optimum time.

Max Iter: 10
Water content tolerance: 0.001 (0.002 for faster run)
Pressure head tolerance: 1 cm (2.5 cm)
Adjust interpolation tables around values you expect (i.e., 0.1 cm to 5 m)

Jirka


**ID = 343, Scale values**

Jodofe, 02/16/2005
Hi, I would like to compare the matrix potential of two soil profiles with the same scale values, but i can't because there i don't know how to change the values of the scale. It is posible, if yes, how can i do it.

Thank you
Jose Doerner
Mirek, 02/16/2005
Hi Jose:
I'm not very sure what "scale values" you mean but if you want to set your own values for the color scale then it is possible. The method is described for example in
http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=217
(paragraph #3 of Jirka's reply).

You should have the latest version (2.102) if you want to have this option working properly.
Mirek


**ID = 344, Point source flow**

Nasserga, 02/24/2005
Dear Prof. Jirka hi
I have a 2D domain (1mX1mX1m) ,how can I represent a point source water supply such as drip point with water flux density(qw=1cm/hr) in both surface and subsurface conditions?

Thanks
Dr. Gamal Khalil


Jirka, 02/26/2005
Send me an email address to which I can send you such example. But the email system must allow larger files than yahoo. At least 1 Mb.
J.


**ID = 345, Evaporation with depth**

Kgreaser, 02/25/2005
Is there a way to get information on Evaporation with depth? I need to find out the effective extinction depth of evaporation, ie the depth below which evap has little to no effect. So far I can't find that information in any of the output files. Information in evap per node would be ideal.
I would appreciate any suggestions anyone has.
Kelly


Jirka 02/25/2005
Evaporation is only through the soil surface. Thus it is not destributed with depth. That's done only for transpiration. If you want to see how deep evaporation affects pressure heads and water contents, just specify some observation nodes with depth and check the results.
Jirka


Absaibes,  04/26/2005
Does this mean that the evaporation simulated by Hydrus 1D is not the cumultative evaporation from all the profil? Is it only the evaporation from the surface layer?
Thanks for answering
Aline

Jirka 04/26/2005
Evaaporation is from the soil surface. Obviously, as a result of this (evapiration from the surface), water needs to flow in the profile towards the soil surface (due to the pressure head gradient). Thus the process of evaporation does affect pressure head (water content) distribution in the large part of the profile.
Jirka


Wibri,  04/27/2005
Is there any evaporation from the soil below the surface? I would have thought the soil atmosphere below surface saturated. Just wondering?


**ID = 346, Input data**

Kaka,  02/25/2005
Hi,
I downloaded HYDRUS 1D recently. I am going to use it for infiltration simulation. Now I have some questions. To enter meteorological data, how it can be exported to the model? There is options for meteo data at 'Time Information' box but it is off (non-active). From where it does load for example rainfall data? Thanks for your attention in these matters.

Regards
Kaka


Jirka 02/26/2005
Kaka,
Present version (2.x) does not allow the input of meteorological data (radiation, wind speed, air temperatures, and relative humidity). One has to directly enter precipitations, and potential values of evaporation and transpiration. Some new features that we are currently working on for the next version (3.0) now already appear in the graphical interface, but are disabled.

Jirka


**ID = 347, Solute transport**

Ihssan,  02/26/2005
Sir,
I am studying solute transport in the unsaturated soil zone (column study). My question is :
How can I enter the effect of pH,pollutant concentration,EC, and other chemical factors on the simulating of solute transport.

thank you, and with my best wishes
ihssan

Jirka, 02/27/2005
Answer is simple. You can not.
J.S.


## ID = 348, Colloid transport

Felipe, 02/27/2005
Dear group,
Im a new user of the program who is just beginning to read the manual and before going further I would like to ask the experts if it is possible to use the program for colloid transport analyses using sand columns. Please be patient with me since I haven't got an engineering background. I am a formed biologist who is studying bacterial transport through porous media (biocolloids). I'm very interested on quantifying the influence of various biological factors on the mobility of colloids. In a nutshell Im not so interested in predicting or modelling this transport but on quantifying the actual changes on colloid transport parameters as affected by the biological parameters (i.e. analysis of break through curves, collision efficiencies, transport velocities, colloid retention, remobilization, etc). Is the program useful for this purpose or only for solute transport?

Thanks a lot in advance
Felipe Leon


Ntoride, 03/08/2005
Dear Felipe:
I am sorry for my slow response. CXTFIT is based on analytical solutions
for the convection dispersion equation (CDE), and the nonequilibrium CDE
(the mobile immobile model, the kinetic adsorption model) as describe in
the manual. As you know well, colloid transport in soils is very
complicated, obviously much more complicated than the assumptions for
the equilibrium and/or nonequilibrium CDE. However, it is sometimes
useful to apply the simple assumption to describe the transport even if
the reality is more complicated than the assumption. What I would like
to say is that it depends on the purpose. It is important to understand
the model assumptions and the limitation of these assumptions. It is
generally necessary to use numerical calculations for more complicated
models. If any further questions, please do not hesitate to ask me.
Regards,
Nobuo
Jirka 03/08/2005
Felipe,
In about two month time we plan to release a new version of HYDRUS-1D that will have some of the features available that you are asking about, such as straining, attachment/detachment, immobilization, filtration, collision efficiencies, etc. Check this site from time to time for news.
Jirka

felipe 03/29/2005
That sounds good, thanks a lot for you answer,
I will be checking periodically,
Felipe.


Jirka 03/29/2005
We want to release the new version before April 24.
Jirka


**ID = 349, Quick questions on Hydrus 2D approach**

sjbmodel 03/03/2005
Hello All,
I have to complete a quick turnaround on a 2-D cross-sectional model of a leak from a lined water reservoir. I plan to simulate either 2 or 3 soil types (from bottom to top - sandy silty clay, a sand, and a sandy silt). The objective is to use an estimate of how far the leaking water has migrated laterally away from the reservoir assuming properties for the above soil types and a leakage rate. Although there is a lot of head to drive water out the actual leak in the reservoir liner, the leak rate may be less than the Ksat of the soil adjacent to the leak due to various factors. I need to include a cross-sectional area of approximately 10 meters vertical and 100 meters horizontal.

My questions are:
1) I have Hydrus 2D vers 2.008, any specific advantages to getting a current version for my problem?
2) Any suggestions in making my model development, running and presentation of output for such a relatively large area more efficient?
I have a lot of experience with unsat modeling just not Hydrus 2D. Thank you in advance for any suggestions

Steve


valliyappant 03/03/2005
Hi Steve,
1. All the latest versions are free to download from the website www.hydrus2d.com provided you have authorisation for your current 2.008 version. You can also get more details about the changes, developments and the advantages of each hydrus2d versions therein. The version updates are mainly related to fixing the errors or improvements. Basically the numerical strategy,working process, etc are all same.

2. Since you want to estimate only the lateral migration of water flow for a cross-sectional area of approximately 10 meters vertical and 100 meters horizontal,

a. Choose horizontal flow in the geometry dialog box.

b. The mass balance error is one of the approximate method to check the accuracy of the numerical results. So by adjusting the spacial and temporal discretization try and keep the error less than 1%.

c. For the total simulation time you may need to go for trial and error. Because you do not know the time for lateral migration of water flow for approximately 100 meters in horizontal direction.

NOTE: WAIT FOR JIRKA TO REPLY
valliyappan.t

Valliyappant, 04/12/2005
A simple question to the forum; which I didn't note till now. I used windows to create backup copies for my files; which is a bit time consuming compared to 'SAVE AS' option in FILE MENU.
How does the 'SAVE AS' option in FILE MENU works? It doesn't work like windows based application I suppose. It makes a copy of the pre-processing data's alone. Is it right?

Is it same in all Hydrus2d versions?
Thanks for your time and help.

Jirka, 04/12/2005
Valliyappan,
You can make complete copies (providing that names of the project are shorter than 9 characters) of Hydrus projects using the "Project Manager".
if you open a project and then use the command "Save as", the code saves only the input.
You can make copies of Hydrus project also using Windows tools (e.g., Explorer). In that case you need to make copies of both project folder ("project_mame") and the "project_name.h2d" file.

Jirka
Valliyappant, 04/15/2005
A question to the Hydrus2d forum.
I got an error message when I tried to open the 'Graphical display of results' dialog box; which is as follows:

INVALID FILE FORMAT!
FILE: C:\HYDRUS2D\PRACTICE1\FLUX.SLOP\H.OUT
I am getting this message only when i create new folder, then workspace and run the project for the first time in that new workspace.

I am not getting this message all the time. I suppose once in 50 times or something like that.

what could be the reason for this? Any suggestions would be great.
THANKS FOR YOUR TIME AND HELP.

**ID = 350, Input doesn't equal the output in balance.out?**

steesk-8 03/03/2005
Hi,
I'm modelling the transport of chloride around a road construction and I've had lot's of problems with a non-moving solute plume.

But right now I'm just trying to compare my inputs with the output in balance.out.I started by running a simplified version of the model with time variabel rain,temperature and solute that runs over period of 1705 days. In this case I'm using atmospheric BC as upper boundary all other boundaries are No flux.
My question is why doesnt totalprecipitation*width equal Volume(final time)-Volume(Time=0), in balance.out. In my case totalprecipitation*width is equal to 10*(Volume(final time)-Volume(Time=0)). The error for the solute i about the same. Where did the rest of the water and solute go?
Total precipation = tot infiltr = tot vAtm = tot rAtm.
Tot runoff = Tot evapor =0.
(WatBalR= .381%, CncBalR=7.348% & Peclet= 1.874 )?(I know that CncBalR is to big.)
I managed the do these types of calculations before with more basic models.

What is the output in cMean(j) in Solutex.out? It seems to be the mean solute fluxes across diffrent boundaries and the total solute flux across the atmospheric BC.

/Stefan


Jirka, 03/03/2005
Stefan,
The code calculates the error in the mass balance by checking the change in storage against fluxes over boundaries. Since your reported mass balance error for water is only 0.381% then the solution must be OK, and whatever infiltrated into the transport domain must indeed resulted into the change in storage.

In solutex.out, I report actual and cumulative solute fluxes over particular boundaries. cMean is the solute flux divided by the water flux, and thus is the average concentration. THis is not a regular feature of Hydrus and thus is not seen from the graphical interface and is only in the output file.

Jirka
**ID = 351, About the point source flow**

Nasserga, 03/03/2005
My Dear Prof. Jirka
Thank you for your reply about my question and my e-mail is:
nasserga@yahoo.com
my present mail is:
Dr. Gamal Khalil

Soil Science Dept.
King Saud University
P. O. Box 2460
Riyadh 11451
Saudi Arabia

Please if you have any papers concerning this point,send me it
thank you in advance and please accept my personal regards.
Dr. Gamal Khalil


## ID = 352, Orthomin

mizo316, 03/04/2005
Hi Jirka -
I am running a fine mesh (about 3 cm size of an element) for a 100 m wide by 10 m deep domain. I am getting the message "Orthomin terminates /reduce time step"

I am using the regular tolerance values, I have decreased my time step and still same message. I realize that my time step should be favorable with the discretization, so I wonder if this message has other meanings. Does reducing the time step really helps? Or I should give up on that mesh? Or I should keep doing trial and error for tolerance and time step parameters.

P.S. for a mesh which is about 10 cm for an element, the simulation runs fine. But I found that for the coarser mesh, the migration of liquid is way underestimated. Injected liquid is traveling much faster for the finer mesh, which is closer to the "true field case".

Thanks
Mazen

Jirka ,
03/04/2005
ORTHOMIN is an iterative solver that solves the matrix of linear equations. It almost alwaya converges. However, if the mesh is too small then the solution may diverge. In the older versions of Hydrus, I would indeed end simulation whenever this happened. However, I changed that few years ago and do allow Hydrus to reduce the time step itself, and you should get first the message:
"ORTHOMIN terminates - too many iterations, Time step reduced"
and the code before terminating tries smaller time steps. It terminates only when it hits dtMin.

Jirka


mizo316, 03/18/2005
Hey Jirka --

I am not sure why the solver is not converging for such a reasonably small mesh size (3-5 cm). Is there any way to go around this problem without increasing the mesh size? Would a faster computer help?

Please give me your opinion on this issue.
Thanks,
Mazen


## ID = 353, Run time and ways to speed up

Sjbmodel, 03/07/2005
Jirka,
I would not think that the problem I am posing is that numerically challenging nor that poorly posed but it seems to be running exceptionally slow. Only 0.3 day of simulation time completed in almost 18 hrs of run time. Here is the problem posed:
-a 5m wide by 10m tall 2-D vertical cross-section modeled with a rectangular grid with 2 cm square elements (i.e., 250 cols and 500 rows)
- BCs are no flow on right side, atmospheric on top, free drainage on bottom, 0.5 m of constant pressure (= +50 cm) in upper portion of left side, with the remainder of left boundary being no flow.
- initial condition = -100 cm everywhere
- tinit and tmin = 1e-6 d, tmax = 0.25 d
- material properties equal to silty sand values provided in Hydrus
I am running this on a fairly slow PC (P3, 1 Ghz) but it seems to be running way slower than this problem should. Any ideas on the problem? Thanks.


Valliyappant, 03/07/2005
I did have the same problem for my simulation as well. There are lots of topics in this forum which are related to reducing the simulation time.
you can refer to the following topics to get better idea

1. Simulation time; Topic ID 342
2. Model running time; Topic ID 317
3. Long running time; Topic ID 150

These are few guidence values given by Jirka in regard to reducing simulation time. This is for a specific problem.

Theta tolerance: 0.001
Head tolerance: 1 cm
Initial time step: 0.01 d
Minimum allowed time step: 1e-6 d
Maximum allowed time step: 1 d
Discretization close to the surface: 1 cm
Discretization at larger depths: 5 cm (not on boundaries between layers).
If you have clay soils, use -2 cm air entry option.
hCritA= 150-500 m
If you have course soils, decrease hCritA to less than 100 m.

I think you need to choose 'General Option' in the geometry dialog box and create your own mesh using meshgen. (Try this once you get your results and mass balance error information).

Jirka, 03/07/2005
You have 125 thousand nodes. That's a pritty big problem. Do you need such a fine grid. I would assume that you could optimize the grid. The problem as you describe it seems manageable.

Jirka

Jicerman, 04/06/2005
I am also having a runtime problem. I am trying to simulate a situation similar to the "infiltration from a subsurface source in a vertical plane" example, but Ive moved the source to the surface and changed the soil to sand. Similar mesh design except again the concentratin has been moved to the surface. I have no problem running the simulation with a loam making me think the problem is with the sand hydraulic properties...
Ive tried your suggestions from previous post, such as changing the time steps, pressure head and water content tolerances, all yielding little success.
Are there any other suggestions perhaps specific to sandy soils?
Thanks much.

Jirka, 04/06/2005
Sand has much more nonlinear soil hydraulic properties than loam. As a result the moisture front is much sharper. Sands typically require much finer discretization than loams.
Jirka

**ID = 354, Question about cvalue1,cValue2 and cValue3**

Azhdary 03/14/2005
Hello Jirka
I have almost completed my study thanks for your suggestion. However I have 2 questions.

1. First one is related with cValue parameter in Time Variable Boundary Condition editor. In this I am giving cValue2 as 1. But I have worked out solute flux as 0.38 mg/ml, I would like to know the value 1 for cValue2 is correct or not. I checked the help file and found the dimension of cValue as ML-3. In one of the your papers (published in Agricultural Watwer Management Journal 2005)as coauther you have taken it as relative concentration. If it is relative concentration then it may be dimensionless. Please advise on this issue.

2. question 2. In same your paper the out put of the Hydrus-2D (contor plots of relative nitrate concentration for Drip)has been presented along with values of X and Y scales. In our case we are not getting values of X and Y scales. We are using Hydrus-2D Version 2.102. How can we get the values of X and Y scales?

Thanking you
Khalil

Jirka 03/16/2005
Khalil,
1. cValue (and other concentrations) can have either concentration units (mass/volume) or it can be dimensionless. If you look at the governing convection-dispersion equation, you can easily see that the whole equation can be divided by c0 and the results will then be in dimensionless concentrations.
2. My coauthors did the graphs in Surfer, where they added x and y scales.

Jirka

Azhdary, 03/28/2005
Jirka
Thanks for your reply, after your advise my simultion improved but still I have 2 problems.

1. In your Agricultural Water Management paper(2005), in Table No. 3 changing the fertigation strategy resulted in a change of %N leaching. However in my simulation(surface drip fertigation)with changing the fertigation strategy, suppose from Irrigation-fertigation to Fertigation- irrigation in the end of first irrigation cycle and end of crop season %N leached of total N added do not changes. Please advise me how can I solve this problem.

2. In Time Variable Boundary Condition editor changing the value of cValue2 from default value to 1 or zero or any other value the same result is coming. Kindly advise me how to solve this problem?
Best regards
Khalil

Jirka, 03/28/2005
Khalil,
In the Boundary module you need to specify the solute transport boundary condition. When you select Cauchy BC, you will be asked for a pointer to the vector of boundary conditions. If you use cValue2 to specify concentration, then this pointer must be equal to 2.

Jirka

**ID = 355, Overland flow**

wcorneli 03/15/2005
hi,
in studying subsurface flow and runoff on a 100 m long hillslope, we are thinking to apply Hydrus2D. we know that thera are plans to include overland flow into Hydrus2D and are therefore wondering if this will be done in the near future. if not, is there at this time a beta version available that could be tested? actually we have a dataset with pressure potentials measured at 10 depths and at 10 locations along the hill.
thanks for your reply.
wim

Jirka 03/17/2005
Wim,
I have been working on implementing such option into Hydrus. But at present I share it only with my collaborators and do not plan to release it to the general public until we apply it ourselves first to some available dataset, test it properly, and publish it. At this moment the coupling of the Richards equation (subsurface flow) with a kinematic equation (overland flow) is rather unstable and computations are time consuming. I would assume that I'll make it available in about a year.

Regards,
Jirka

Wcorneli, 01/11/2007
Dear Jirka,
we were wondering if you plan a soon release of the coupled overland-subsurface flow version of Hydrus (kinematic eq coupled to Richards eq)?
Kind regards,
Wim

Jirka, 01/11/2007
Wim,
Not at the current time. Nothing changed since my last posting. I have not worked on this topic. As I said above, I'm willing to share what I have, but do not plan to release it in the general version (which has assumed higher level of stability, correctness, error-free, ...., than my other versions).
J.

**ID = 356, Drainage**

mizo316, 03/20/2005
Between seepage face and free drainage, which boundary is more suitable to simulate subsurface drainage pipes?
Thx
Mazen


Valliyappant, 03/21/2005
Hi Mazen,
I would go for seepage face boundary condition to simulate subsurface drainage pipes.


Jirka 03/21/2005
Mazen,
Free drainage is certainly not suitable for simulating flow to subsurface drainage pipes. Free drainage is only a vertical flow due to gravity.
Use seepage face. That's how drains operate. There is no flow as long as the soil around the drain is unsaturated. The flow starts with saturation. And that's exactly what seepage face BC does.

Jirka


valliyappant 03/23/2005
Dear Jirka,
I did understand what a free drainage and deep drainage does. I would like to know under what circumstances (For example: seepage face BC for subsurface drainage pipes) each of the above BC's are more suitable for practical applicability? One practical applicability example for each would be great (in words; not a simulation).

Thanks for your time and help.


Jirka, 03/23/2005
Free drainage - this is ua nit gradient boundary condition that should be used at the bottom of the soil profile where groundwater level is very deep and thus does not affect flow in the transport domain. This boundary condition considers simply only gravitational flow at the bottom of the soil profile.

Deep Drainage - this BC relates flow at the bottom of the soil profile to the position of the groundwater (that groundwater must be above the bottom of the soil profile, i.e., within the transport domain). There is a special function that relates this deep recharge with the position of the groundwater level. This function should predict that deep recharge is larger when groundwater is high, and smaller when it is deep. You need to have information about this relationship before you can use it.

Seepage face - this BC should be used on the side of dikes (where you expect that water may leave the transport domain), on the bottom of lysimeters, or for drains. This BC states that there is no flow as long as boundary is unsaturated, and flow starts

only with saturation. At saturation, boundary head is set to zero and the code calculates the flux.

I hope that this is helpful.
Jirka


## ID = 357, Deuterium transport

Tine, 03/21/2005
I am interested in the transport of Deuterium through the unsaturated zone. I tried to simulate the solute transport with Deuterium values with ranges of -20 to -80 per mil. But it didn´t work. Do I have to enter to Deuterium in per mil or in ppm? I only have weekly values, but I want to calculate in daily time steps. Do I have to repeat the weekly Deuterium values each day?
Perhaps somebody can help me. Thanks


Jirka  04/05/2005
Hydrus will allow you to use any mass units (dimensionless, ppm, mol, mg, ....) since the governing transport equation is unit-independent.
You can specify boundary values at any time interval as well. The code then chooses the optimum time step.
Jirka


Tine, 04/05/2005
My problem was that my Deuterium values are negativ. But now I use the absolute value and it perfectly works.
But is it possible to get a mean flow velocity of tracer or the mean transit time?
Thanks a lot!


Jirka, 04/05/2005
Hydrus reports Darcy's fluxes. To get pore velocities, just divide that with water contents. The code does not report residence times. That you need to figure out from the breakthrough curves at the input and output.

Jirka


## ID = 358, Infiltration during raining event

lgowdish 03/22/2005
I wanted to test a model that I wrote for infiltration and redistribution against a known model, so I chose Hydrus 1D. I have been trying to run a couple of trials and have not been successful in getting them to run - I keep getting 'numerical solution not

converging'. I think I have entered in the inputs properly, but there are a couple that I am unsure of:

1) Using the Brooks and Corey model (I chose this one because that is the one that I could find all the inputs I needed for my model) - the manual says that the input for l is the pore connectivity parameter and that it was originally equal to 2. However when I am looking at sand, it shows it as a 1. What is the correct input? Is there an equation for determining it?

2) I have rain inputs which I enter into the variable bc page, but what do I enter in for HCritA? In the manual it says it needs to be determined from equilibrium conditions between soil water and atmospheric water vapor, but what is the equation?

3) I want to simulate rain events, have no ponding, but allow for surface runoff, so as my upper bc do I use - atm. bc with surface runoff?

I also realize that since this is a numerical model, the time inputs are critical for proper convergence. I have tried numerous values for initial time step, minimum time step, maximum time step, lower/upper optimum range and lower/upper time step multiplication factor with still no convergence (sometimes most time steps do not converge and other times only the beginning part of the simulation does not converge). Is there a way to determine what the best values for these inputs are?

Any help would be greatly appreciated.

lgowdish 03/23/2005
I have just read through a lot of the topics in the Hydrus 2-D discussion group and found the answer to my question regarding HCritA. I then re-ran my sand trial using a HCritA of 10000cm and I am still not able to get it to converge. I ran a very similar trial with clay (the difference being the hydraulic parameters and the rain rate) and got what looks like good results (they matched up with what I got in my model as well as the results that another paper got). Since I was able to get clay to converge and get good results, I am trying to figure out what I need to change in the sand trial to get it to converge. In another discussion topic someone said they were having a problem convergin because of a really high rainfall input and/or abrupt change in rainfall. My simulation has a rain input of:

0.25 50 cm/h
3.0 0
3.25 50
6.0 0

Could this be causing a problem? My other thought was the time step but I am using a minimum time step of 0.0001 hr, which I thought should be okay. I decided to try another HCritA and see what happens for sand. When I tried 1000cm I got it to converge, and have a good mass balance error, but all it was doing was having all the rain go to runoff and not infiltrate. Then I lowered the HCritA to 50cm and I got it to converge, but my mass balance error was high (96%). So what would be the appropriate HCritA and/or what else would I need to change?

Thanks. Leslie

Jirka, 03/23/2005
Hi,
1) I've formulated in Hydrus Brooks and Corey functions slightly differently than it is in RETC (from which I took the formulation for Brooks and Corey functions).
In RETC:
K=Ks*S^[L+1+2/n]

In Hydrus:
K=Ks*S^[L+2+2/n].

Thus L should be 2 in RETC and 1 in Hydrus.

2) hCritA is similar to the wilting point. And thus it should be somewhere below -150 m. It can be calculated from relative humidity as follows:
Hr=exp[h*M*g/R/T]
where Hr is the relative humidity [-], h is the pressure head [m], M is the molecular weight of water, g is the gravitational acceleration, R is the gas constant, and T is temperature [K].

3) Correct

Hydrus almost always runs with default values. For infiltration, you can decrease the initial and minimum time steps to around 1 s.

Jirka


Lgowdish, 03/24/2005
I am trying to simulate the infiltration and redistribution of a couple rain events. I have only been able to get one of the trials (clay) to converge. I have adjusted some of my inputs as a result of the advice from Jirka, but I am still having a problem. One thing I have noticed is that if I reduce the rain intensity so that all the rain from at least the first rain event goes to infiltration (ie. no ponding or runoff) then I can get it to converge. As soon as I increase the intensity so that there is runoff during the first rain event, it does not converge. Does anyone know why that is and/or how I can get it to converge at the higher intensity (note: I have already tried reducing the initial and minimum time step)? Thanks.


Jirka 03/24/2005
USe van Genuchten's functions and you will have no problem with convergence.
If you use Brooks and Corey you need to either disable internal interpolation tables or adjust them to the air-entry pressure head value.

Jirka

lgowdish 03/24/2005
Thanks Jirka. Disabling the interpolation tables has worked for most of the trials. I am still having a problem with sand, but I will try using van Genuchten's functions for that one. Alpha for brooks and corey is the inverse of the bubbling pressure and n is the pore distribution index, but how do I determine the alpha and n for the van Genuchten function?
Thanks again.


Jirka, 03/24/2005
In both functions, the parameters alpha and n are basically a fitting parameters reflecting the shape of the retention function. There has been attempts to assign the physical meaning to these parameters, but in my view they both are fitting parameters.

There are obviously many method how to get them, e.g.:
a) Fitting retention data
b) Pedotransfer functions
c) Parameter estimation by calibrating numerical model against transient experiments,
....
Jirka


## ID = 359, Organic layers in Hydrus

steesk-8, 03/24/2005
Hi
I´m simulating solute transport from a road. Right now my soil model only consists of two coarse soil layers with grass on top, but I would
like to have some sort of organic(humus)layer on top of the two coarse layers. I´ve been having trouble finding data for organic soil layers, so I wonder if anyone has some good data for such layer or other ideas how too model this.

thanks
stefan


## ID = 360, Questions of MESHGEN2D and Initial Condition

mzk1973_82, 04/03/2005
Dear Sir
This is Babu from Iwate University, Japan. I am a first time user of HYDRUS 2D. I am trying to simulate the water movement through a heterogeneous slopping soil profile resulting from a one-day rainfall. The model domain is around 32.44m in width and 3.50m depths at uphill and 3.00m depth at down hill with a natural gentle slope of 5.6 degree. Three materials with three distinguished vertical layer at 0.25, 1.55, 1.70m at uphill and .21, 1.55 and 1.24 m at downhill. We have W.C readings by TDR at 5, 10, 20, 30, 40, 60, 80, 100, 120, 140, 170, 200, 250 and 300 cm for both up

and down hill. Now I have some specific questions on MESHGEN and Initial condition as follows:

1) Is it possible to generate 5cm x 5cm mesh in this unorthogonal domain? If possible would you kindly let me know the process sequentially? Here, I would like to mention that I have tried to generate 10cm x 10cm mesh, but it seems to me that it is not working properly.

2) For initial condition I want to assign WC, which is measured from the field. I want to check the linear distribution with depth. In that case do I need to assign angle for each layers or only for surface layer? Moreover, can I use 5cm and 25 wc values at top and bottom values for first 25 cm layer and so on for the next two layers? Although I am not using the down 300cm field measured values in this case.

3) If I want to compare the observed results with the measured one then I need to set the Observation nodes on 5, 10, 20, 25,  c  c.300cm. I tried to set the obs. Nodes on these points but failed to do so. How can I do that again and what are things I need to take care to set obs. Nodes on these points?

Thank you very much for your patient hearing. I am really sorry to bother you. As I am beginner I will highly appreciate your cordial cooperation in this regard.
Sincerely yours
Babu, Iwate University, Japan


Jirka, 04/05/2005
Babu,
1. For this problem I would use the internal mesh generator for rectangular domain. Play with it a little bit and you will see that you can descrive your transport domain using input parameters, such as slope of the domain and dz values. I would recommend to use large discretization in the horizontal than vertical direction and more nodes closer to the surface than in depth (use lower upper densities).

2. Linear distribution with depth can be used only for pressure heads (if I remember correctly). Other than then, Hydrus gives you complete freedom in specifying the initial conditions. I would, however, recommend to always use initial conditions in terms of the pressure head. If you have measured water contents, then you can recalculate them into pressure heads using retention curve.

3. You should be able to specify observation nodes in any node of the finite element grid (spatial discretization). If you have problem doing so, go the options->Grid setting and disable the grid.

I hope this was useful.
Jirka
**ID = 361, Can I use Hydrus 1D to simulate air entrapment?**

Soilj, 04/04/2005
I doubts if I can use HYDRUS 1D to simulate air entrapment in soil column.
Thanks

Jirka, 04/05/2005
You can. You need to use hysteresis model.
Jirka

## ID = 362, Fibreglass wick lysimeter

SCHNEIDER, 04/07/2005
Hello,
I am Sebastien, a M.Sc. student in soil science at INRA Grignon.
I would like to model the behavior of fiberglass wick lysimeters in soil. Our lysimeters are 25 X 25 cm and are placed at 40 cm depth.
The wick length (altitude difference between the lysimeter plate and the bottom end of the wick) is 70 cm.
I tried to put a fixed potential boundary condition at the lysimeter plate, but it feeds the surrounding soil with water when the soil matric head is lower than -70 cm. If I put a seepage condition, then the lysimeter flows out only when the soil is saturated over the lysimeter.
I guess I need to implement a seepage face that could allow water to enter the lysimeter as soon as the matric head exceeds -70 cm.
Am I right ? and if yes, how could I do that ?
I guess some people must have been faced with this issue already...

Thanks for helping
Sebastien

Jirka, 04/07/2005
Sebastien,
Send me an email and I will send you a version that initiates seepage face with -70 cm pressure head.
Jirka

## ID = 363, Problems with 2 soil material simulation

Trombalac, 04/19/2005 :  23:16:42
Hi, im`modelling a flux of water throught a column of soil that in the surface has layer of Clay of 30cm, and later a layer of Sand of 170cm depth.
I choose 2 soil types and 2 soil layers for mass balance. My BC are surface runoff as upper BC and free drainage as lower BC. The initial conditions are water content with 7% for clay and 4.6% for Sand.
The problem is when i run the program, hydrus not converge in pressure and a value in the column of v/KsTop made ****** in the change of soil type and soil layer. (the same node)
Why this happened? How can i deal in the best way with 2 types of soil?

Jirka, 04/19/2005
When you have clay in the system, select the model with 2 cm air-entry value.
I would assume that your initial condition is highly at nonequilibrium. The pressure in the clay is probably hundreds of meter, while in sand less than a meter. That may be rather unrealistic.

Jirka


Trombalac, 04/20/2005
Jirka, you have right. It`s a enormous differnce of pressure. I choose air entry value, however, i still have the problem with the discontinuity. I fixed that changing the wrong value in the out.files, because there be only 2 nodes that presents problems.
Ok, thanks for your answer.


Absaibes, 04/27/2005
I'm facing the same problem of discontinuity at the interface between different soil layers. The modeled pressure head (and consequently the water content ) at the nodes of the interface show a clear cut or jump. This jump is sometimes greater than 5m from node to node knowing that each node represents a depth of 1 mm.
1-is there any solution to smoothen this radical jump in pressure head?
2-Does this discontinuity affect the simulated fluxes (especially evaporation)?
NB: my soil materials vary from loam to slightly compacted loamy sand to sandy loam.

Thanks for answering
Aline


Jirka, 04/27/2005
I would expect large differences in water content at the interface. However, pressure heads must be continuous. There may obviously be different pressure head gradients in different materials. Since the conductivity will be distinctly different in different materials, then to produce the same flux you need different pressure head gradients in different materials.
Jirka


## ID = 364, Governing transport equation

steesk-8, 04/21/2005
Hi
I am simulating single-ion transport in a road-structure and I have some questions about equation 3.21 in the Hydrus-2D manual.If your solute only exists in liquid phase and doesn't react at all. If you have no adsorption, no decay, no uptake of solute by plants. Is F=0 and G=0. Or is F= S and G= -Sc if so do Fc+G=0.
I don't think I understand the definition of F and G.

Thank you
stefan


rvang 04/21/2005
Stefan:
Those two terms lump all first-order degradation (F) and zero-order production (G) processes in your problem, including root uptake (S) if present (in G). They are all zero in your case (i.e., all gamma's and mu's are zero). In addition your retardation factor may be 1 (no sorption or vapor phase partitioning). --Rien van G.


## ID = 365, Version 3.0 of Hydrus 1D

Jirka, 04/21/2005
Dear colleagues and friends,
We have released a new version of the HYDRUS-1D, version 3.0. Feel free to download this version, together with the updated manual, from our download section. This new version (3.0) has the following new features:

1) Root water uptake with compensation
2) Additional analytical models for the soil hydraulic properties suggested by Kosugi [1996] (log normal model) and Durner [1994] (dual porosity model)
3) Water flow in the dual-porosity system
4) Solute transport with attachment/detachment coefficients, permitting simulations of colloid, virus, and bacteria transport
5) Two kinetic sorption sites (one can be used for example for the air-water interface)
6) Filtration theory based evaluation of attachment coefficients
7) Carbon dioxide production and transport module
8) Geochemical carbonate chemistry module that considers transport, precipitation/dissolution, cation exchange, and complexation reactions for major ions.
9) The new model is (may be) about 3 times faster than the old model.

Please, keep up informed if you encounter any problems with this new version. Although it was extensively tested, there may obviously be some bugs that escaped our attention. Let us know about it. Let us also know what you think about these new features and what other processes you would like to see in the software package.

Also, please, help us in spreading the news about this new release.

Best regards,
Jirka, Rien, and Mirek
(HYDRUS development team)

## ID = 366, CXTFIT

Sharendr, 04/25/2005
Dear group!

I am modelling some laboratory experiments using CXTFIT model. I don't have any idea about what the equation you have used and what are the initial and boundary conditions. Could you tell me what is the equation you used and the variables you used. Is it possible to model the results using porevolume vs concentration?

Where can I get the following book?
Determining transport parameters from laboratory and field experiments Bull 84-3 Va Agric.Exp.St.Blaacksburg, Va, 1984.

what is the beta term you refer to in CXTFIT model? What do you mean by stochastic problem?
harendra

Jirka, 04/25/2005
Are you using our STANMOD model, which includes CXTFIT2. Then you should look at the manual of CXTFIT. The manual gives all the governing equations, as well as it explains all the terms. I will email you the CXTFIT manual.

Best regards,
Jirka

Sharendr, 04/25/2005
Hi Jirka!
Thank you very much for your information. I am using STANMOD model. My email address is ps_harendra@yahoo.com. Please email the CXTFIT manual.
Rgds
HArendra

Jirka 04/25/2005
Your email account is too small for this pdf file. Download the manual at:
http://www.ussl.ars.usda.gov/models/cxtfit.HTM
Jirka

Sharendr, 04/26/2005
Hi Jirka!
In CXTFIT model. what do you mean by flux averaged concentration and resident concentration?What are the equations used for calculation of above mentioned concentration. Since I am collecting effluent at the outlet, how to calculate flux averaged concentration and resident concnetration?
Rgds
Harendra

Jirka, 04/26/2005
Simply: If you collect effluent then it is flux concentration. If you extract a sample and measure concentration in it, that is the residual concentration.

See the paper referenced below on the discussion of flux and resident concentrations.
Van Genuchten, M. Th., and J.C. Parker. 1984. Boundary conditions
for displacement experiments through short laboratory soil
columns. Soil Sci. Soc. Am. J. 48:703-708.

sharendr 04/26/2005
Dear Jirka!
Thank you very much for your help. I am doing my experiment in 75 cm long and 5 cm diameter column. I am pumping constant concentration of solution through top of the column for 3 minutes. And collecting the effluent ar periodic time intervals from the bottom.Is it a semifinite column or finite column? What is the best boundary condition for the following experiment.Is it a step input or pulse input?

Rgds
HArendra

Jirka, 04/26/2005
Pulse input
Semiinfinite column.
J.

sharendr 04/28/2005
Hi Jirka!
When I use CXTFIT model, I am facing some problems. Could you please answer me the following questions?
1) In semiinfinite column, what is the boundary condition (third type, flux averaged concentration) physically represent?
2)In flux average concentration profile, the STANMOD package produces some graphs according to spatial difference? What does it mean? Is it the cross section of the column at different intervals?
3)How can I measure R, D values by just collecting the effluent from the column?
4)Since I don't know the parameters of R,D, how can I fit the data in inverse problems?
5)What is the difference between one site and two site region model?

Thank you for your help.
Rgds
Harendra

sharendr 05/03/2005
Dear Group!
Could anyone of you help me to the following equations?

When I use CXTFIT model, I am facing some problems. Could you please answer me the following questions?
1) In semiinfinite column, what is the boundary condition (third type, flux averaged concentration) physically represent?
2)In flux average concentration profile, the STANMOD package produces some graphs according to spatial difference? What does it mean? Is it the cross section of the column at different intervals?
3)How can I measure R, D values by just collecting the effluent from the column?
4)Since I don't know the parameters of R,D, how can I fit the data in inverse problems?
5)What is the difference between one site and two site region model?

Thank you for your help.
Rgds
Harendra

Ntoride, 05/08/2005
Harendra:
> 1) In semiinfinite column, what is the boundary condition (third type,
> flux averaged concentration) physically represent?

Please see p.4-5 of the CXTFIT manual. The tird-type BC is generally
preferred except for a very small water flux condition (diffusion is
dominant at the boundary). Strictly speaking, the column experiment is
a finite system. The solution for a semi-infinite system can be used for
a finite system when the bottom boundary does not affect the upstream
solute transport. Furthermore, the flux-average concentration subject to
the tird-type BC for a semi-infinte system is a kind of adjustment to
represent a effluent concentration for a finite soil column.

> 2)In flux average concentration profile, the STANMOD package produces
> some graphs according to spatial difference? What does it mean?

The flux concentration is defined as
$c_f = c_r - D/v \, dc_r/dx$ (Eq. 2.13)
The solute flux is given by $v*c_f$.

When you calculate $c_f$ vs. depth, the $c_f$ profile represent the solute
flux profile (v is a constant value).

> 3)How can I measure R, D values by just collecting the effluent from the
> column?
> 4)Since I don't know the parameters of R,D, how can I fit the data in
> inverse problems?

R can be determined by the difference between a nonreactive tracer and
a reactive solute. D is generally determined using a nonreactive tracer
such as Cl assuming R = 1. The D value for a tracer could be used for
a reactive solute.

> 5)What is the difference between one site and two site region model?

Please see p.15-17 of the manual and references in the manual.The two
site model becomes the one site model when f = 0.

If any further questions, please let me know. Please also carefully read
the CXTFIT manual as well as some of the references in the manual.

Nobuo
Sharendr, 05/09/2005
Hi Nobou!
Thank you very much for your answers. Now it is easy for me to continue.
Rgds
Harendra

Sharendr, 05/13/2005
Hi group!
Could you tell me, how can I measure D (Dispersion) in tracer studies in lab scale
column experiments?

Ntoride, 05/18/2005 :  11:44:40
Harendra:
Many people are using CXTFIT to estimate D and/or other parameters from
breakthrough curves (or concentration profiles). You can have these
information from various textbooks such as Methods of soil analysis
(SSSA book series No.5). Please note that this forum is not designed to
have fundamental information. I am willing to discuss with you when you
encounter specific problems with CXTFIT.

Nobuo

**ID = 367, Flux data in observation nodes**

s.zanarello 04/27/2005
Hi
I'm using Hydrus-1D to simulate water flux within an unsatured profile. I'd like to
obtain flux vs. time at various observation nodes within the profile (not only at profile
bottom).
Is it possible to obtain this output with Hydrus-1D?
Thanks

Jirka 04/27/2005

You have this information in the profile information, i.e., in the nod_inf.out file. However, this information is printed only at print times.

Jirka


s.zanarello 04/28/2005
Thanks Jirka for your answer
So, it's not possible to obtain the flux vs. time for every time step for observation nodes (not only for print times). Is it true?


Jirka 04/29/2005
No, it is not in the standard Hydrus code.
I will email you a version in which I print fluxes in observation nodes instead of temperature into the obsnod.out file.
Jirka


s.zanarello 05/02/2005
Thank you very much Jirka
you are very kind.
Regards


## ID = 368 Transport of Heavy Metals

Norman, 05/04/2005
Hi Rien and Jirka,
I have a question to ask you or anybody in the forum. Can HYDRUS be used to model precipitation/dissolution and redox reactions for heavy metals? If not, do you know any oher vadose zone models that can do that?
Thank you for your help.
Noman


Noman,
We will release in about a month a program called HP1. This is HYDRUS coupled with PHREEQC. Below is a brief description of this program. I do not know about anything better!!! :-) If you email me I can send you references mention in the text below.

Jirka


## HP1
HYDRUS-1D was recently coupled also with the PHREEQC geochemical code (Parkhurst and Appelo, 1999) to create a new comprehensive simulation tool, HP1 (acronym for HYDRUS1D-PHREEQC) (Jacques et al., 2003; Jacques and Šimùnek, 2005; Šimùnek et al., 2005). This new code contains modules simulating (1) transient

water flow in variably-saturated media, (2) the transport of multiple components, (3) mixed equilibrium/kinetic biogeochemical reactions, and (4) heat transport. HP1 is a significant expansion of the individual HYDRUS-1D and PHREEQC programs by preserving most of their original features and capabilities. The code still uses the Richards equation for simulating variably-saturated water flow and advection-dispersion type equations for heat and solute transport. However, the loosely program can now simulate also a broad range of low-temperature biogeochemical reactions in water, the vadose zone and in ground water systems, including interactions with minerals, gases, exchangers, and sorption surfaces, based on thermodynamic equilibrium, kinetics, or mixed equilibrium-kinetic reactions.

Jacques et al. (2003) and Jacques and Šimùnek (2005) demonstrated the versatility of HP1 on several examples such as a) the transport of heavy metals (Zn2+, Pb2+, and Cd2+) subject to multiple cation exchange reactions, b) transport with mineral dissolution of amorphous SiO2 and gibbsite (Al(OH)3), c) heavy metal transport in a medium with a pH-dependent cation exchange complex, d) infiltration of a hyperalkaline solution in a clay sample (this example considers kinetic precipitation-dissolution of kaolinite, illite, quartz, calcite, dolomite, gypsum, hydrotalcite, and sepiolite), e) long-term transient flow and transport of major cations (Na+, K+, Ca2+, and Mg2+) and heavy metals (Cd2+, Zn2+, and Pb2+) in a soil profile, f) cadmium leaching in acid sandy soils, g) radionuclide transport, and h) the fate and subsurface transport of explosives.


## ID = 369, Random field

mizo316 05/07/2005
Hi Jirka -
When running a simulation for a generated random hydralic conductivity field, is there a way to check the see number of the realization? and is there a way to see what was the correlation lengths used? It seems that once you run the program, you cannot go back and check what values you inputed!

Thanks --
Mazen


Jirka 05/07/2005
Mazon,
That is correct. I do not save parameters of the random field in input files. Parameters are entered in the Boundary module, and the random field is generated directly in this module. Since it is a random field, this field can not be generated again anyway (only fields with the same statistical properties). It is a responsibility of the user to remember (or write down) used parameters for these random fields.
You can obviously see the generated values of scaling factors (and thus also calculate back their statistics if the domain is large enough) in the input file Domain.Dat (once you save it as ASCI).
Jirka
 mizo316 05/11/2005
Hi Jirka --

What are the names (or the headers) of columns that present the parameters of the scaling factors in Domain.Dat?
Thanks,
Mazen


Jirka, 05/11/2005
Axz, Bxz, and Dxz for scaling factors for pressure heads, conductivities, and water contents, respectively.
Jirka


**ID = 370, Continue a run**

mizo316, 05/11/2005
Hi Jirka--
Is it feasible to run a simulation having initial conditions (ex. water content) as the output of a previous file? or simulation?

Thanks -
Mazen


Jirka,  05/11/2005
Mazen,
Sure. HYDRUS-2D allows to import results from previous simulations as initial condition for new runs - see Import Pressure (water contetns, concentrations) in Boundary module. You can also check out our web page here for tutorials (the last one for 2D).

Jirka


mizo316, 06/16/2005
Hey Jirka –
I used importing pressure head for one of my simulations. It ran fine. In the output, i wasn't able to get through obesrvation nodes, water boundary flux, and such from the output menu.
I find that strange, does that often happen or there is something wrong with this particular simulation.
Thanks -
Mazen


Jirka, 06/16/2005 :
Just close HYDRUS and reopen it. You should get the graphs. HYDRUS and WINDOWS sometime loose pathway to the definition of the graph.
J.
**ID = 371, Water balance**

mizo316, 05/11/2005
Hi Jirka--
Is it feasible to run a simulation having initial conditions (ex. water content) as the output of a previous file? or simulation?
Thanks -
Mazen


Jirka, 05/11/2005
Mazen,
Sure. HYDRUS-2D allows to import results from previous simulations as initial condition for new runs - see Import Pressure (water contetns, concentrations) in Boundary module. You can also check out our web page here for tutorials (the last one for 2D).

Jirka


mizo316 06/16/2005
Hey Jirka --
I used importing pressure head for one of my simulations. It ran fine. In the output, i wasn't able to get through obesrvation nodes, water boundary flux, and such from the output menu.
I find that strange, does that often happen or there is something wrong with this particular simulation.

Thanks -
Mazen


Jirka, 06/16/2005
Just close HYDRUS and reopen it. You should get the graphs. HYDRUS and WINDOWS sometime loose pathway to the definition of the graph.
J.


**ID = 372, Importing steady-state flow to be initial cond.**

Marek, 05/15/2005 :  20:32:05
Using Hydrus2D I ran steady state flow simulations to establish the flow field for a subsequent flow and solute transport simulation, and I have encountered several problems as listed below:

1. Using water content as the top (0.1) and the bottom (0.43) boundaries, steady-state conditions are obtained with the mesh 38x44 and 75x87. When using a denser mesh 149x173, i.e. 0.125 m (for the domain 18.5 m x 21.5 m that includes 4 different soils) the steady state conditions can not be found. The message reads "orthomian terminate too many iterations time step reduced". What could be the problem with the denser mesh that I thought I might need for a subsequent solute transport simulation?

2. When the steady-state solution (of grid 75x87) is imported (using "copy" in Project Manager) to become the initial conditions for the subsequent solute transport simulation, the calculations can not be completed even before entering solute transport parameters. Just running a steady state simulating with the imported initial conditions, the DOS window appears and disappears before it can be read and a window shows up with the message: "an error log is being generated, OK", but the log is nowhere to be found.

Needless to say, entering the new time dependent flow boundary (precipitation), and solute transport parameters and their boundaries also produce incomplete calculations and the same message as said above.

I would appreciate your assistance.
Cheers,
Marek


Jirka, 05/16/2005
Marek,
If the code can not find steady-state solution (unchecked water flow) in one step (which I actually would be surprised if it could for a problem with four materials and initial conditions given in terms of water content), try to find the steady solution simply using time stepping, i.e., running the problem for a long time until the pressure heads (or water content) stop changing.

Although Hydrus allows specifying initial and boundary conditions in terms of water contents, I do not recommend using such option, especially if there are more soil materials involved. It then often happens that the problem is physically unrealistic, with huge pressure head gradients (after water content are converted) between different materials. I would always recommend using pressure heads (which is the driving force for flow).

Jirka


Marek, 05/16/2005
Thanks Jirka!
The soil water content is known to me, but not the pressure. Are you suggesting that I should use water retention curve for the given soil to assign pressure head at the top of the soil column to convert known water content to the pressure value?
Thanks again.
Marek


Jirka, 05/16/2005 :
Yes, that's what I would do. J.


Preecha, 11/03/2005
Hi Jirka,

I do not quite understand how changing initial conditions between water content and pressure head can affect the predictions, if HYDRUS-2D uses water content tolerance for unsaturated conditions and pressure head tolerance for saturated conditions.
Is it the initial condition or the convergence criterion that may lead to inaccurate pressure head profile?

Kind regards,
Preecha

Jirka, 11/03/2005
Preecha,
The results should be identical whether you specify the initial conditions in terms of the water content or pressure head. The reason that I always recommend to specify the initial conditions in terms of the pressure head is that you can see immediately that your initial conditions make sense. If you do that in water content, that may not always be the case. Imagine that you have a three-layered profile and that you specify the initial conditions in terms of measured water content. Hydrus then converts these water contents values into pressure heads using the retention curve. It often happens that pressure heads in different layers are then order of magnitude different, which obviously does not make sense. This is result of the combination of errors in measured water contents and specified retention curve. I guess that you would never specified such erroneous initial conditions using pressure heads.

Jirka

**ID = 373, Saturated boundary and solute transport**

Marek, 05/16/2005
With the "Atmospheric" boundary conditions (rain only) in the top of the soil column and the "Constant pressure" flow boundary (full saturation) in the bottom of the column to simulate water table, the "Tthird type" solute transport boundary shows up automatically at both ends of the column (and can not be removed).

This also happens in the example "fluxinf" if the lower flow boundary is changed from "No flux" to "Constant pressure" (0 value) to simulate saturated conditions at the bottom of the column. The results of the simulation show two plumes developing at both ends of the column and converging in the center of the column.

Obviously, I am doing something wrong with my boundary assumptions for simulating downward vertical flow with the spill of contaminant at the top surface to be driven down by atmospheric precipitation. Could you help please?
Cheers,
Marek

Jirka, 05/16/2005
There must be boundary condition specified at every boundary where water flow boundary is specified, i.e., where water can cross the boundary. The code needs to

know what to do with the solute there. Thus you can not remove solute transport boundary condition (since that would be wrong), but you can certainly change it. You can also specify values for each BC. Thus you get plume flowing through the boundary only if water flows in and you specify concentration in it.

As a default we always use Cauchy boundary condition, that automatically is changed to Neuman condition, if the flow is out of the domain.

Jirka


## ID = 374, File size

mizo316, 05/19/2005
Hi Jirka --
For an output file (example: Obsnod.out), is there a way to reduce the size of the generated time columns? Is that dependant on the minimum time step used for the input? Please let me know if there is a way to go around it without producing tens of thousands of rows.

Thanks,
Mazen


Jirka, 05/19/2005
In the curent version you can only select output at specified print times or at every time step. In the version 3.0 of HYDRUS-1D I have implemented additional options for output management:
a) print at every n time steps
b) print at regular time interval (e.g. every day).
I will have these option into HYDRUS-2D update that will be (hopefully) released in the fall (as part of HYDRUS-3D).

Jirka


mizo316, 05/20/2005
Hi Jirka --
If we have that option in the current Hydrus-2D version, please let me know how to select output at specified print times.

Thanks,
Mazen


## ID = 375, Graphical display requires reentering the programme

Marek, 05/23/2005

Every time I run HYDRUS-2D and then, using Graphical Display, set the parameter to display and set the range of nodes for the diagram, the diagram does not show up unless I exit the program, open it again and repeat the procedure. My operation system is Windows 2000 and I am using HYDRUS-2D Version 2.102. Is it the problem with installation or slight incompatibility with Windows 2000?
It is not a big deal to do so, but to certain extent it is annoying.
Cheers!


Mirek, 05/23/2005
Hi Marek,
I remember we had a similar problem in the past that was fixed in June 2002 in version 2.007: Fixed a problem in module Graphics, 1D-Graph dialogs. After exporting data to a file in directory different than the Hydrus installation directory the 1D-Graph dialog didn't show up anymore. It was necessary to restart Hydrus-2D to make 1D-Graph working again.

Unfortunately I am not able to reproduce your problem in version 2.102 on my computer. Please could you describe exactly (step by step) what you do before the 1D-Chart stops working? The most easy way is to use the "Print Screen" button to copy all your steps to a *.doc file and then send me the file to mireks@pc-progress.cz

Thanks
Mirek


Marek, 05/23/2005
Hi Mirek,
Well, I hit "enter" after a successful program run; double click on Graphical Display result; in Grid setting set the grid, check snap to grid and grid visible; check icon "values along the cross section; click on the first node of the profile; click on the last node of the profile. And nothing is showing up in my window unless I go through the same procedure after reentering the program. Fortunately it is not a big deal but annoying. I will send you the procedure via Print Screen, as you advised, later.
Thanks for your interest.
Marek


Mirek, 05/23/2005
Marek:
you need not send me the pictures, now you have decribed it clearly enough. I think that this is the same error as we fixed in 2002: some command changes "current directory" of the Hydrus2D and then the 1D-Chart can not find file "CHART.VTC" (that is in the Hydrus installation directory). The current directory can be changed typically by "Open File" or "Save As" dialogs but these commands in Hydrus2D have special parameters not to change the current directory. I think that in your case the current directory is changed before you run the Graphics module - I'm going to look at the code once more. I'll inform you later.

Just one more remark: please check your Hydrus-2D shortcut on desktop and make sure that the field "Start in directory" contains the path to the Hydrus-2D installation directory.

M.


Mirek, 05/24/2005

Marek:

I've checked Hydrus source code and I think that the problem (1D-Graph dialog doesn't show up) can occur only if you import initial conditions from an *.out file before the calculation. Is it your case?

I'll fix this error in a next version.

Mirek


Marek, 05/24/2005

You bet I did!

As a matter of fact I ran my 30-year simulation in 4-year pieces importing the flow field from an earlier run. Thank you for your help and consistency in tracking the bug despite that I tried to down play its importance. As you could see from my subsequent posting I had a bigger problem caused by myself and solved by Jirka who "read my mind". You guys are great! Thanks!

Marek


Koenverbist, 10/20/2005

As a first time user I followed the examples in the handbook "Modelling Variably Saturated Flow with Hydrus-2D".

Following these examples until p. 1.45 the same error occurred, and the 1-D graphs could not be shown. Indeed, one of the previous examples shows the use of the h.out file for importing initial conditions and this seems to cause an error.

Therefore, I am glad this problem was mentioned here as a subject and I really think others might experience the same problem.


**ID = 376, No solute in the domain**

Marek, 05/23/2005

Hi Jirka,

Here I am again. I followed your advices (thank you!) to run my 4-layer model with initial conditions using pressure instead the water content. Then I applied atmospheric boundary at the top of the soil column. This atmospheric boundary was 24 precipitation events a year, each lasting 0.5 days and reduced to 20% of the value to compensate for lack of evapotranspiration data. I ran this scenario for 30 years (each year having the same distribution and values of precipitation) i.e. until there was no change in water pressure distribution along the 21 m soil profile above the water table. Having this well established flow field I introduced solute transport; i.e. entered

Kd, dispersivity, bulk density, defined duration of the pulse as 1 day, have cBdn1 listed "1", entered positive value of concentration (10000) in the column cVal1-1 for the one day period that was followed by the 24 precipitation events (same as for establishing the flow field), selected Third type solute boundary. Well, after running the program I have neither concentration in the soil profile diagram, nor I have the solute fluxes for any 7.5-day print-interval through one year period.

May you kindly help me again by telling me what I am missing in my procedure?

Thanks,

Marek


Jirka, 05/23/2005

Solute must be entered on the same day as you have water flux (precipitation) into the profile. The third BC is equal to q*c, where q is water flux and c is concentration, and thus you need both values to be nonzero to get solute flux into the profile.

Also make sure that solute BC pointer in the boundary module is equal to 1 (if you use cValue1).


Jirka


Marek, 05/23/2005

Thank you Jirka for the promptness and the clear explanation!

It is so obvious that I feel embarrassed not thinking about it myself. Yes, I entered my 1-day spill time 5 days before I had first precipitation event.

I also appreciate your remark about solute BC pointer to be equal to 1, but at least hear I did not make a mistake (or the program did not allow me to make it).

Regards,

Marek


**ID = 377, Hydrus changes initial conditions**


Eric, 05/23/2005

Hello,

I am running a simulation of infiltration into a dry (or nearly so) soil. I assign a uniform initial pressure condition (=-300 cm throughout the domain). Though I assign -300 cm as the initial pressure, once the model starts running it appears that HYDRUS changes the initial condition to ~ -250 cm (this is shown in both the graphical display of results and the observation node data). I would guess that the reason HYDRUS2D does this has something to do with the fact that the initial pressure condition I assign yields an initial moisture content very near to zero. Is this correct?

For reference, the retention parameters I am using are given below. I realize that these are not typical of many natural soils. This is primarily a modeling exercise, not a "real-world" simulation.

ThetaR=0

ThetaS=.3

alpha = 0.04

n=9

Thanks for any information you can provide regarding the cause of my mysterious changing initial conditions.
Eric


Jirka, 05/23/2005
Eric,
For the numerical reasons, the model requires that the initial condition has saturation at least 0.00000001. For your soil hydraulic parameters, initial saturation of 0.00000001 leads to the pressure head of only -250 cm. Smaller saturation would lead to conductivities, which can not be handle with any available precisions provided by numerical models.
Jirka


Eric, 05/23/2005
Jirka,
Thanks for your quick reply. Now that I understand how Hydrus deals with this problem I will be able to modify my scenario to work within the numerical constraints.
Eric


## ID = 378, Floating point error with solute transport

Heartblood
Hello all!
I'm trying to model solute transport, and everytime I set a BC as "constant flux" the program won't calculate, exiting with a floating point error.

My aim is to see how solutes from a landfill infiltrate into groundwater. The thing is that I must have constant fluxes for the groundwater.

Can anyone help me?
Thanks in advance, Joachim
PS.: I already tried to set the time criteria to a smaller value, but it didn't help


## ID = 379, Groundwater flow under a landfill

Heartblood, 05/27/2005
Hello,
I want to simulate polution comming out of a landfill under which groundwater flows in a certain direction.

Is there any way that one can build a mesh with another mesh inside?

Everytime when I build a mesh and create another one inside that mesh the second one cuts kind of a hole into the other one. It's just like having some kind of obstacle in a flux that blocks the flow.
When simulating the result is that no water is flowing below the landfill.

Thank you,
Joachim


Jirka, 05/27/2005
Joachim,
When designing transport domain, you use internal curves. These internal curves, if closed, are by default treated as holes in the transport domain. You need to change the boundary type. Do that as follows (all in the boundary module):
a) View->Edit Boundary Points
b) Edit->Boundary Points->Change Boundary Type
c) Click on the internal Curve
d) In the dialog that shows up, change the type from "Hole" to "Internal Curve"

Then this will be part of the transport domain.
Jirka


Heartblood, 05/30/2005
Well, I might be stupid, but I can't find any
View -> Boundary Points etc.
Edit -> Boundary Points -> Change Boundary Type
etc.
Where exactly should that be located in the program? Is it in the Boundary Conditions Editor?
Thanks again


Mirek, 05/30/2005
Joachim:
the function is not in the "Boundary Conditions editor", it is in "Geometry and FE-Mesh" editor. Start this module and then follow Jirka's instructions.
Mirek


Heartblood, 05/30/2005
Well, well, monday morning
I've found everything now, thanks for the help!!
But one problem still remains, when I change the curve into "Internal Curve" I can not add any boundary conditions in the Boundary Editor.
Is there any way to have both options together? I mean, have the flow region without any holes AND the possibility to add boundary conditions.

Joachim
Jirka, 05/30/2005

Boundary conditions can be specified only on boundaries, not in internal curves that are part of the domain.
J.

s.zanarello, 06/08/2005
Hi Heartblood
just a question about your problem (Jirka, excuse me...): why do you want to use Hydrus-2D for simulation of groundwater pollution from landfill? It's a typical groundwater problem and, although Hydrus is a great software for unsaturated zone, it's not the most widely used and accepted software for groundwater flow and transport problems.
I suggest Modflow+MT3D, the most widely used software for groundwater; you can use commercial version (es. Visual Modflow) or Processing Modflow (a little bit less friendly than Visual Modflow...but freeware).
By

rvang 06/08/2005
Interesting comment. Well.... HYDRUS may not be used as much (yet) as MDFLOW/MT3D but maybe it should, even for saturated only flow/transport problems. Also, many landfill situations involve variably-saturated (vadose zone/groundwater) transport scenarios. HYDRUS includes many, many other features that are not included in the other codes. So, think again! --Rien van G.

## ID = 382, Fine particles

Sharendr, 05/31/2005
Hi!
1)Can I use the STANMOD package for fime metal particle ( colloidal) particles through soil medium?
2)What are the limitations of STANMOD package?

harendra

Jirka, 05/31/2005
STANMOD
Software package
Evaluating Solute Transport in Porous Media Using Analytical Solutions of the Convection-Dispersion Equation

STANMOD (STudio of ANalytical MODels) is a Windows based computer software package for evaluating solute transport in porous media using analytical solutions of the convection-dispersion solute transport equation. Version 1.0 of STANMOD includes the following models for one-dimensional transport problems: CXTFIT 2.0 [Toride et al., 1995], CFITM [van Genuchten, 1980], CFITIM [van Genuchten, 1981], and CHAIN [van Genuchten, 1985]. Version 2.0 of STANMOD, to be released in the

spring of 2000, will also include the models 3DADE [Leij and Bradford, 1994] and N3DADE [Leij and Toride, 1997] for two- and three-dimensional transport problems.

CXTFIT

The software package includes a modified and updated version of the CXTFIT code of Toride et al. [1995] for estimating solute transport parameters using a nonlinear least-squares parameter optimization method. This code may be used to solve the inverse problem by fitting a variety of analytical solutions of theoretical transport models, based upon the one-dimensional convection-dispersion equation (CDE), to experimental results. The program may also be used to solve the direct or forward problem to determine concentrations as a function of time and/or position. Three different one-dimensional transport models are considered: (i) the conventional equilibrium CDE; (ii) the chemical and physical nonequilibrium CDEs; and (iii) a stochastic stream tube model based upon the local-scale equilibrium or nonequilibrium CDE.

CFITM

STANMOD also comes with an updated version of the CFITM code of van Genuchten [1980] for analyzing observed column effluent data using analytical solutions of the one-dimensional equilibrium convective-dispersive transport equations. The code considers analytical solutions for both semi-finite and finite columns. The model provides an easy to use, efficient and accurate means of determining various transport parameters by optimizing observed column effluent data. CFITM represents a simple alternative to the much more comprehensive, but also more complex, CXTFIT model.

CFITIM

STANMOD also contains an updated version of the CFITIM code of van Genuchten [1981] for analyzing observed column effluent data using analytical solutions of the one-dimensional equilibrium and nonequilibrium convective-dispersive transport equations. The code involves analytical solutions for semi-finite columns. The nonequilibrium solutions consider the two-region dual-porosity (bi-continuum) flow model for physical nonequilibrium and the one-site or two-site sorption models for chemical nonequilibrium. The model provides an easy to use, efficient and accurate means of determining various transport parameters by optimizing column effluent data.

CHAIN

In addition, STANMOD 1.0 includes the modified and updated CHAIN code of van Genuchten [1985] for analyzing the convective-dispersive transport of solutes involved in sequential first-order decay reactions. Examples are the migration of radionuclides in which the chain members form first-order decay reactions, and the simultaneous movement of various interacting nitrogen or organic species.

3DADE

STANMOD 2.0 will include the 3DADE code of Leij and Bradford [1994] for evaluating analytical solutions for two- and three-dimensional equilibrium solute transport in the subsurface. The analytical solutions assume steady unidirectional water flow in porous media having uniform flow and transport properties. The transport equation contains terms accounting for solute movement by convection and

dispersion, as well as for solute retardation, first-order decay, and zero-order production. The 3DADE code can be used to solve the direct problem, i.e., the concentration is calculated as a function of time and space for specified model parameters, and the indirect (inverse) problem in which the program estimates selected transport parameters by fitting one of the analytical solutions to specified experimental data.

N3DADE
Finally, STANMOD 2.0 will incorporate the N3DADE code of Leij and Toride [1997] for evaluating analytical solutions two- and three-dimensional nonequilibrium solute transport in porous media. The analytical solutions pertain to multi-dimensional solute transport during steady unidirectional water flow in porous media in systems of semi-infinite length in the longitudinal direction, and of infinite length in the transverse direction. The solutions can be applied also to one- and two-dimensional problems. The flow and transport properties of the medium are again assumed to be macroscopically uniform. Nonequilibrium solute transfer can occur between two domains in either the liquid phase (physical nonequilibrium) or the absorbed phase (chemical nonequilibrium). The transport equation contains terms accounting for solute movement by advection and dispersion, as well as for solute retardation, first-order decay, and zero-order production.

Test Examples distributed with the model:
STANMOD is installed with numerous examples that are divided into eight groups (workspaces): CFITM, CFITIM, CHAIN, Direct, Inverse, Stochast, 3DADE, and N3DADE. The first three workspaces CFITM, CFITIM, and CHAIN contain examples of the corresponding models. The next three workspaces Direct, Inverse, and Stochast contain examples of the direct, inverse, and stochastic problems solved with the CXTFIT model. The last two workspaces 3DADE and N3DADE contain examples of the 3DADE and N3DADE models. Users are advised to select an example closest to their particular problem, copy this example and then simply modify the input data.

Examples of the CFITM model
1. The transport of chromium through a sand, a semi-infinite system.
2. The transport of chromium through a sand, a finite system.

Examples of the CFITIM model
1. A generated data set using the assumption of physical non-equilibrium solute transport
2. The movement of tritiated water through a Glendale clay loam in a 30-cm long column. The nonequilibrium transport model with five fitted parameters (P, R, $f\grave{O}$, $f\ddot{A}$, and the dimensionless pulse time, T0).
3. The movement of tritiated water through a Glendale clay loam in a 30-cm long column. The linear equilibrium adsorption model with three fitted parameters (P, R, and T0) was used.

Examples of the CHAIN model
1. The transport of the three-species nitrification chain (NH4+ - NH2- - NH3-).
2. The transport of the radionuclide decay chain (238Pu - 234U - 230Th - 226Ra).

Examples of the CXTFIT model:

a) Direct problems:

1. The first-order physical nonequilibrium model to calculate concentrations in the mobile and immobile phases as a function of time at a depth of 50 cm for a 2-d pulse input at the soil surface. Four different combinations of the mobile water content ($ƒÒ$ = $ƒám/ƒá$ = 0.25, 0.5, 0.75, and 0.99) and the transfer rate ($ƒç$ = 0.2, 1, 5, 1000) are used

2. Ditto with five different fractions of sorption sites (f = 0, 0.25, 0.5, 0.75, 1) that equilibrate with the mobile region are used.

3. Example illustrating the effect of the first-order decay constant, $ƒÝ$ (=0, 0.25, 0.5, 1 d-1) on solute distribution.

4. Example calculatiing flux (cf) concentrations for two values of the Peclet number, P (= 2, 10) as a function of relative distance when solute-free water is applied to a soil having a stepwise initial resident distribution.

5. Ditto for and resident (cr) concentration

6. Example showing the effects of the first-order mass transfer rate coefficient, $ƒÑƒn$(= 0.08, 0.2, 10, 1000 d-1), on breakthrough curves in terms of the flux-averaged concentration, as a result of applying a Dirac delta input function to an initially solute free soil.

7. Example calculating breakthrough curves according to the two-site nonequilibrium CDE for four values of the fraction of sorption sites (f = 0, 0.3, 0.7, and 0.999). Value of 0.08 for the first-order mass transfer rate, $ƒÑƒzƒn$were used.

8. Ditto with the value of and 0.2 d-1 for the first-order mass transfer rate, $ƒÑƒzƒn$were used.

9. Examples calculating breakthrough curves using different sets of R, $ƒÑ$, and f values in the two-site nonequilibrium CDE for Dirac delta input and pulse input, respectively.

10. Example involving a direct problem using deterministic nonequilibrium transport as described by an initial value problem. The case calculates equilibrium and nonequilibrium resident concentrations and total concentration profiles at T = 1 for three values of the partitioning coefficient $ƒÒ$ (= 0.1, 0.5, and 0.9).

11. Examples demonstrating the effect of the first-order decay coefficient $ƒÝ$ (= 0, 0.1, 0.2, 0.35) on Picloram movement through Norge loam using either a pulse or step input, respectively, and assuming applicability of the equilibrium CDE.

12. Example demonstrating the effect of the dimensionless mass transfer coefficient $ƒç$ (= 0.001, 0.28, 0.7, 1.7, 2.8, 7.0, 1000000) on calculated effluent curves for 2,4,5-T movement through Glendale clay loam using the two-region physical nonequilibrium CDE model).

13. Example demonstrating the effect of the dimensionless partitioning coefficient $ƒÒ$ (= 0.2, 0.35, 0.5, 0.65, 0.80, 0.99) on calculated effluent curves from, and spatial concentration distributions in, an aggregated sorbing medium, respectively, assuming two-region physical nonequilibrium transport.

14. Examples demonstrating the effect of the retardation factor R (= 1.0, 1.75, 2.5, 3.5, and 5.0) on calculated effluent curves from, and spatial concentration distributions in, an aggregated sorbing medium, respectively, again assuming two-region physical nonequilibrium transport.

15. Example calculating solute concentration versus time and distance for an aggregated sorbing medium, respectively, assuming two-region physical nonequilibrium model, as affected by the dimensionless mass transfer coefficient $ƒç$ (= 0.02, 0.2, 0.5, 1.5, 7.5, 1000).

16. Example demonstrating the effect of the Peclet number P (= 5, 15, 40, 100, 10000) on calculated effluent curves from, and spatial concentration distribution in, an aggregated sorbing medium, respectively, assuming two-region physical nonequilibrium transport.

b) Stochastic problems:

1. Examples calculating field-scale resident concentrations, cr, versus depth resulting from the instantaneous application of a solute to the surface as a BVP (variable mass) and an IVP (constant mass).

2. Examples calculating field-scale resident concentrations versus depth as a result of a pulse-type solute application of constant duration.

3. Example demonstrating the effect of variability in the pore-water velocity, v, on the field-scale resident concentration profile and the distribution of the variance for cr in the horizontal plane. This example calculates the mean resident concentration and its variance as a function of depth at t = 3 d for three values of ƒãv (=0.1, 0.3, 0.5) as a result of a 2-d solute application to a solute-free soil.

4. Examples calculating the breakthrough curves for three types of field-scale concentration modes.

5. Example demonstrating the effect of correlation (ƒâvKd = -1, 0, +1) between the pore-water velocity, v, and the distribution coefficient, Kd, on calculated field-scale resident concentration, cr, profiles. Field-scale concentrations at t = 5 d resulting from a Dirac delta input at t = 0 are calculated versus depth for either perfect or no correlation between v and Kd.

6. Examples calculating field-scale resident and total concentrations, respectively, assuming stochastic nonequilibrium solute transport. Examples assume a negatively correlated v and Kd (ƒâvKd = -1) using three values of the mass transfer coefficient ƒç.

c) Inverse problems:

1) The pore-water velocity, v, and dispersion coefficient, D, are estimated from breakthrough curves measured at three different depths (11, 17, and 23 cm) with four-electrode electric conductivity sensors. Breakthrough curves were a result of (a) continuous application of a 0.001 M NaCl solution to an initially solute-free saturated sand, and (b) leaching with solute free water during unsaturated conditions, respectively.

2) Two examples of nonequilibrium solute transport consider transport of tritiated water and boron, respectively, through Glendale clay loam in a 30-cm long column. In both examples parameters of the non-equilibrium transport model were optimized against effluent curves.

3) The stochastic option of CXTFIT, together with parameter estimation, is demonstrated with two examples. The first example pertains to resident concentrations in a 0.64-ha field to which a bromide pulse was applied for 1.69 d followed by leaching with solute-free water [Jury et al., 1982]. The stream tube model was used to estimate the mean pore-water velocity <v>, the mean dispersion coefficient <D>, and their standard deviations ƒãv and ƒãD, respectively. The second example demonstrates the estimation of parameters in the stream tube model for reactive transport using a hypothetical data set. The standard deviation, ƒãKd, and the coefficient of correlation between v, and Kd, i.e., ƒâvKd, were fitted to the hypothetical data, while keeping <v>, ƒãv, and <Kd> constant.

Examples of the 3DADE model

1) Example calculating steady-state concentration profiles for a diffuse solute source in one quadrant of the soil surface.

2) Examples calculating transient concentration profiles for transport from a rectangular solute source at the surface using either a first- or third-type boundary condition.

3) Example calculating transient concentration profiles for transport from a parallelepipedal initial distribution.

4) Example calculating transient concentration profiles for transport from a circular solute source at the surface using a third-type boundary condition.

5) Example that considers (similarly to Example1) solute application in one quadrant of the soil surface. The parameters R, Dx, and Dy (retardation factor, and dispersion coefficients in the x- and y-directions, respectively) are fitted using a breakthrough curve at a specified position and the steady-state profile at a selected transect.

6) Example involving the estimation of the parameters R, ƒÝƒzƒnƒÜƒzƒnDx, Dy, and Dz (retardation factor, first-order rate coefficient for decay, zero-order rate coefficient for production, and dispersion coefficients in the x-, y- and z-directions, respectively) for solute transport from a parallelepipedal initial distribution. Breakthrough curves at ten positions along the x coordinate and two transverse profiles were used for the problem.

7) Example that concerns the application of a solute pulse from a circular area at the soil surface. Parameters t0, Dx, and Dr (pulse time, and dispersion coefficients in the x- and r-directions, respectively) were estimated using concentrations at several spatial locations at a specific time.


Examples of the N3DADE model

1) Examples calculating breakthrough curves at a depth of 50 cm and the flux-averaged spatial concentration distribution for instantaneous solute application from a disk having radius of 2.5 cm at the soil surface. The problem involves a circular geometry.

2) Example that pertains to flux-averaged concentration profiles resulting from the continuous application of solute to a rectangular surface area (-2.5 < y < 2.5, -2.5 < z < 2.5). It calculates equilibrium, nonequilibrium and total concentrations versus longitudinal distance at three different times, and in the transverse plane at two longitudinal positions.

3) Example that considers an initial value problem (rectangular) with solute initially located in the regions 5 < x < 15 (c = 1) and 25 < x < 35 (c = 0.5) for 15 < y < 25 and -100 < z < 100.

4) Example, which also pertains to an initial value problem (in this case spherical), assumes that the solute has initially a maximum value at the point given by x=5, y=0, and z=5, and that the solute concentration decreases exponentially from the maximum.

5) Example involving solute production in a cylindrical region of the soil (0 < x < 10 and 0 < r < 2.5). The problem is modeled as a production value problem (PVP) with a heaviside function for the longitudinal and transversal directions, with production in the equilibrium phase equal to 0.5, and in the nonequilibrium phase equal to 1.

**ID = 383, Particles**

Sharendr, 05/31/2005
Hi!
Can I consider micro particles (solid ) as solute transport? I am little confusion about solute definition? Can it soild phase. Because I am modelling iron particles transport through soil medium?

harendra

**ID = 384, Debug source code**

Jun, 06/01/2005
Hi,
When I tried to debug the sourcecode from Visiual Fortran, it always stops at the line of "cFileName = 'LEVEL_01.DIR'". I just don't know what does this "level_01.dir" means. Is this a file or anything else? Hope somebody can give me a help.
Thanks so much.

mary 04/18/2006
Hi Jun,
Where you able to solve the problem stated below about bedugging?
Thanks,
J.

Jun, 05/24/2006
I'm not quite sure about your question. But I just deleted the line and linked all the input file myself and it works.

Rayme, 11/06/2006
Hi Jun,
When I tried to debug the sourcecode from Visiual Fortran, it always stops. Would you please tell me how to solve problem with your own way. I am a novice of hydrus. My email is hj_lei2002@163.com. May I have your excuting program by Email if you can. Thanks for your help.

**ID = 385, Precision in CumQ.out with large time series**

Uwe 06/02/2005
Hi,
I have a problem with precision in the CumQ-file.
I calculate daily flux as

day CumQ Flux

7014 9,9994E+01
7015 1,0001E+02 0,0160
7016 1,0002E+02 0,0100

As you can see, precision on day 7015 is 0,001
but from day 7016 on it is 0,01
This happens with large time series - larger than 10000 days...
My first idea was to change the unit from [mm] to [m] to get lower values in the output file - but with no result.
Any ideas? Two questions:
1) How to keep the higher precision of the calculated daily values
2) Has this any effect on the precision of the values in CumQ? Is there a lower precision on the end of long time series?

regards Uwe


Jirka, 06/02/2005
Uwe,
1) The output format in the code specifies printing to the five valid numbers, and thus you can not do anything about it. You can get higher precision during later times (when the contribution of actual fluxes is below precision of 5 valid numbers) if you integrate fluxes from v_mean.out (since obviously cumulative fluxes are actual fluxes integrated over time).

2) This is only precision of the printed output. Internally the precision is given by Fortran Real number, which is 8 valid numbers).

Jirka


**ID = 386, Using root water uptake model with prescribed hea**

Bndiaye, 06/02/2005
Hello everybody
I have actually a problem using the root water uptake module with a prescribed head boundary condition at the soil surface
If any one has experienced such a problem or have an idea on how to solve it please send a reply to this message
Sincerely
Babacar


Jirka, 06/02/2005
Root water uptake is always associated with the atmospheric boundary condition. Specified transpiration rate is in the program multiplied by the length of the atmospheric BC (given in the Boundary.in file) and then redistributed over the root zone (based on specified root uptake intensities). Thus you need to specify

somewhere atmospheric BC (or you need to manually rewrite the lenght of the boundary condition associated with atmospheric BC in the boundary.in file).
Jirka

Bndiaye, 06/09/2005
Hello
I want to thank you Jirka for your reply I would not find it by my self !
Everything is working now!

Yours,
Babacar

## ID = 387, Drip irrigation boundary solute condition

Tillotwr, 06/02/2005
Hello,
I would appreciate guidance calculating the solute concentration corresponding to a time variable boundary condition for a drip irrigation scenario (cValue2). The information is: 1) a solute application event through a drip tape applies X mg per emitter; 2) water flux for a solute application event is Y mm per time (rGWL); 3) diameter of the drip tape is known; and 4) total number of emitters is known. I appear to be underestimating cValue2.

Thanks, Bill

Jirka, 06/02/2005
Bill,
Make the following calculations (I assume 2D domain, and thus everything is per 1 unit length):
$L$ – circumference of the emitter (L)
$q$ – applied boundary velocity at the emitter (L/T)
$Q$ – irrigation flux from the emitter (=$L*q$) (L2/T)
$t$ – duration of irrigation (T)
$V$ – applied volume during one irrigation cycle ($Q*t$) (L2)
$c$ – concentration of the irrigation water (M/L3)
$S$ – solute amount applied during one irrigation cycle ($V*c$) (M/L)
If you know solute amount and want to get concentration, then obviously $c=S/V$
Jirka

## ID = 388, Evaporation

ArnautvanLoon, 06/06/2005
Hello,
I try to simulate rainwater lenses floating on top of seepage, by means of a transport modelling to dinstinghuish different water types. The upper BC is atmospheric, with a

precipitation of 3 mm/day, and an ET of 0 mm/day. The lower boundary condition is a constant flux of 3 mm/day.

Hydrus simulates regularly an outflux over the upper boundary when the seepage reaches the surface level, which means that the EC of the soil profile resembles the presence of seepage water. During two of these timesteps however, the EC locally increases at the surface level (see attachment), which should physically be impossible while evapotranspiration is absent.

Does this error have something to do with the ET=0, or might this be a more numerically problem?

Arnaut van Loon


## ID = 389, Vapor density

Marek, 06/07/2005
Does HYDRUS allow for transport simulation of "heavy" vapor, e.g. PCE (tetrachloroethylene) vapor, downward the soil column via density? If it does allow, what parameters need to be used? I could not find any variable or an equation related to the vapor density, so I guess the answer will be NO, but I would like to be sure.
Thanks,
Marek


Jirka, 06/07/2005
Marek,
You were right. The answer is no. We have no density driven vapor flow. I guess we could implement that if you, or someone else provide some funding.
Jirka


## ID = 390, Monod kinetic equation

Rainfields, 06/09/2005
How to use the Monod kinetic equation to simulate the solute reaction other than first/zero order kinetics?


## ID = 391, Queries on cxtfit 2.0

Amathew, 06/10/2005
Hi!
I am running CXTFIT 2.0 example 7-8b (with MODE=2; NREDU= 1.0). When I run the code for different DT values (Block H), the BTC I get is given for a different ZL(in the output) than what I specified in the input? I thought DT only affected BTC resolution? Would you please help? I should be specifying a value for ZL when the case is mode=2; NREDU=1.0 right?, instead of leaving it blank?

Thanks a bunch! Ann.

Ntoride, 06/11/2005
Ann:
The output format depends on MPRINT in Block K (=1 for conc. vs time, =2 for conc. vs. time, see the manual in p.78). CXTFIT is now in the STANMODE program with a Windows interface. It would be much easier for you to specify parameter values with the interface. You can download the program in this PC-progress site. STANMODE includes all the same examples as in CXTFIT.

Nobuo

## ID = 392, Bottom boundary condition

TIEN DUNG, 06/10/2005
Hello,
I'm simulating a column drainage experiment with bottom condition described such as: water flows from the column bottom through a pipe whose distance between column bottom and exist of outflow is 50 cm.
How do i impose the bottom condition in Hydrus?
Someone tell me, please!
Thank you advance very much.
TD

Jirka, 06/10/2005
If the vertical distance between the bottom of the column and the end of the pipe is 50 cm, then I would used constant head of -50 cm as bottom boundary condition.
Jirka

## ID = 393, How could I get the velocity horizontal?

YANG, 06/13/2005
Hello,
This is my first time to come here ,May I ask you a question? I want to get the velocity horizontal of AB,not the velocity vertical
How can I get it by the 1D-graph?

Valliyappant, 06/13/2005
Hi Yang,
You can get the change in horizontal velocity from the post processing menu.

Step 1: Go to Graphical display of results dialog box
Step 2: Go to 1D graph and choose 'cross section' option
Step 3: Choose any two points (click two times, one to start and one to finish) within the domain in horizontal direction to display the graph with length on x-axis and velocity on y-axis.

If you want to have the flow velocity in horizontal direction, then you need setup a pressure/flux gradient in horizontal direction. This can be achieved by using the the constant Head/Flux boundary.

Hope this helps. I hope you didn't mean horizontal and vertical hydraulic conductivity (anisotropy).

Jirka, 06/13/2005

Yang,

2D graphics displays only magnitude and direction of velocity vectors (fluxes). I does not show horizontal or vertical components. If you want those, you can go to the command "Convert to ASCII" and you will get an ASCII file that has vertical and horizontal components of flux for each node of the the FE mesh.

Jirka

YANG, 06/14/2005

hello,

Jirka,Thank for your help! I find it! thank you very much!

by the way ,I think maybe

the format 5,30648e-007 is better than 5.30648e-007 ,

cause Excel can recognieze 5,30648e-007 as a number. Just a little advice!!

THANK YOU

YANG

Mirek, 06/14/2005

Hi Yang:

just one little remark to the 5,30648e-007 / 5.30648e-007 format problem: The standard format is "." and formatting in Excel is given your local "national settings" on your computer. If you have English Windows then the format is 5.30648e-007 and if you have for example Czech Windows the format is 5,30648e-007. A simple solution is switching your Windows national settings to "English US" for a while when you are reading files with "." formatting and then switch it back.

Mirek

YANG, 06/14/2005

Hello,

Mirek , thank you ,I got it! smile  Merci beaucoup!

## ID = 394, Qr Qs alpha?

YANG, 06/13/2005

Hello:

This is my first time to use this software.some questions

are very simple for you , but for me ,it is difficult.could you
answer me?
For the soil saturé,h>=0,I suppose Ks=1m/s,then how can I treat these left values
Qr=Qs=?,Alpha? n?
Thank for you !


Valliyappant, 06/13/2005
Hi,
This is a very general answer. I hope this helps.
Each type of soil has specific values for the following soil hydraulic parameters Ks,
Qr, Qs, Alpha, n, etc.
You can get the values for the above parameters from Hydrus2d if you know the
1. soil type - Hydrus2d gives list of soils in water flow parameters dialog box.
2. soil texture - Go to neural network prediction
OR
You can get the values for the above parameters from laboratory experiments as well.

WAIT FOR JIRKA TO REPLY.


YANG, 06/13/2005
Hello: valliyappant
firstly,thank you for your answer,but , the question is when I want to simulater the
flow dans the layer of soil satureted ,
so that Qr Residual soil water content should be equal Qs Saturated soil water content,
Qr=Qs, so which values should be added or changed.

thank you
YANG


Jirka, 06/13/2005
Yang,
If you simulate saturated flow and transport, then only two relevant soil hydraulic
parameters are Qs (saturated water content or porosity) and Ks (the saturated
hydraulic conductivity). The other parameters are irrelevant (so you can select any
soil, e.g. loam). Other parameters, i.e., Qr, a, n, l are relevant only for unsaturated
flow.

Jirka


YANG, 06/13/2005
Thank you Jirka,
Yes ,I want to sumulater saturated flow,and I has changed these soil hydraulic
parameters,Qr=0, alpha=0, n=0, l=0,suddenly   Cit is happened a trafic accident. at
least n>=1, but this is for soil saturated. how could I solve this problem?
thanks againt!
YANG

quote:

Yang,

If you simulate saturated flow and transport, then only two relevant soil hydraulic parameters are Qs (saturated water content or porosity) and Ks (the saturated hydraulic conductivity). The other parameters are irrelevant (so you can select any soil, e.g. loam). Other parameters, i.e., Qr, a, n, l are relevant only for unsaturated flow.

Jirka


Jirka, 06/13/2005

As I wrote above, you need to select some soil. The parameters are ignored, but they can not be unrealistic.

J.


YANG, 06/13/2005

ok, I will select it ,par exemple,I choose sand ,just like what you said "The parameters are ignored" , I keep it et do not change it , but after that what is the diffence of the parameter entre the soil saturated et the soil non saturated when I want to simulater these two situations for a samed model.

thank you


**ID = 395, Why invalid file format?**

YANG 06/13/2005

Hello:

Everyone,hello Valliyappant,thank for you answer my question so quickly.

I have another question,when I finished input the B.C "constant pressure" (see this picture).

and after a calulation of HYDRUS-2D,I double click the button Graphical Display Results.it appear that "invalid file format" I dont know why,

thank you

YANG


Valliyappant,  06/13/2005

I did get the same error message a couple of months before. The error message was

INVALID FILE FORMAT!

FILE: C:\HYDRUS2D\PRACTICE1\FLUX.SLOP\H.OUT


I got an error message when I tried to open the 'Graphical display of results' dialog box.

The mistake I did was in the geometry. I got the simulation working. I am not sure whether it is the same for you.

Basically you need to check whether all your input parameters are correct, time and space discretization is adequate, etc.

If you still cannot solve the problem, then open a new project and redo the same problem but be careful and concious when you input each and every parameter.
OR
Send me the file (zip the file please) through e-mail if you want. The e-mail address is valliyappant@yahoo.co.uk


YANG,  06/14/2005
Hello:

Valliyappant,finaly, I find the mistake I did, Cause I didn't choose correctly a type of soil, on the contrary,I change it,so I did get this error message. thank you for your every explications careful.
YANG


## ID = 396, Beginner in need of help!

elduderino3, 06/13/2005
Hello, I am new to using hydrus 1d and i was wondering if there is other literature out there that gives a good BASIC explanation for the "Time Information" in the pre-processing side of the menue? That seems to be giving me the most confusion at this time. I am not a programmer by trade, so the more basic the better. Thanks ahead of time
jordan


## ID = 397, Boundary condition at minerals interface

nwhaws 06/13/2005
Hi all,
How does h2d specify the water and solute boundary conditions at the interface between two different materials (e.g. continuity of concentration, flux etc.)?

Nathan W Haws


Jirka 06/13/2005
Nathan
In H2D materials are assigned to nodes. Therefore there is a diffuse interface between two materials (between two neighbouring nodes withdifferent material numbers). Therefore everything is changing linearly between two nodes (since we use linear finite element scheme), i.e., pressure heads, water contents, concentrations, and temperatures.

Jirka

**ID = 398, Qs and Ks**

YANG, 06/14/2005
hello,
Everyone! Hello,Valliyappant,Jirka and Mirek!
I still have a question.
Could you tell me if I choose a soil type,
for example Sand,flow saturated,
so it coresspond Ks=8,25E-005 ,Qs=0,43
but if I make a test in laboratory, the granule of sand are
very little, so Ks=2,00E-005, Qs=???
Is this parameter Qs obtained by a test hyraulique?

thank you!
YANG GUANG


Valliyappant, 06/14/2005
Hi Yang,

I think the values will be different but should not be in orders of magnitude as you
have mentioned in your topic.
The default values given in hydrus2d is obtained from field tests and they are average
values i suppose. i read it long time back. I need to check again. You can get a clear
idea from the references mentioned.
1.Go to water flow parameters dialog box in pre-processing menu
2.Go to help menu
you will get an idea of the parameters used and from where they have be taken. The
values for the hydraulic parameters are not from laboratory tests i suppose. so there
will be a difference in the numbers used.

I hope this helps.


YANG, 06/15/2005
Hello:
Valliyappant,I have read the references mentioned and use the help menu , but I still
didnt find my answer.It is a pity! However , should say "thank you, Valliyappant!"
YANG


**ID = 399, Boundary conditions???**

YANG, 06/15/2005
Hello:
Everyone, It is me,I want to still ask you a question
please,Could you tell me why?
the question is , Look this picture please ,after I impose "No flux Boundary
Condition" on several boundarys, so I use the button "Constant Pressure" to see the
value of pointA (point A is a point on the boundary of No flux),

his value of Constant Pressure is equal "-1". so I am thirsty for knowing why? It is a value default for this logiciel?

Jirka, 06/15/2005
Dear Yang,
You have so many questions that it seems that we go with you though ever step of the modeling excersise. This site indeed is not meant to explain what the hydraulic conductivity or the saturated water content is. I think the best for you would be, before starting using HYDRUS, to take some basic Soil Physics class or one of our HYDRUS courses (see below). By doing that you can visit some pretty places.

Jirka

June 30 - July 1, 2005
Honolulu, Hawai
The University of Hawaii-Manoa Natural Resources and Environmental Management Department, and Water Resources Research Center are organizing a HYDRUS workshop during June 30 - July 1 2005 following the American Water Resources Association Summer Specialty conference, June 27-29 2005 at the Hyatt Regency Waikiki Resort & Spa, Honolulu, Hawaii. For more details and registration visit http://www.ctahr.hawaii.edu/faresa/Hydrus/

October 17 - 19, 2005
Utrecht University, The Netherlands
This three-day short course and workshop begins with a detailed conceptual and mathematical description of water flow and solute transport processes in the vadose zone, followed by an overview of the use of finite element techniques for solving the governing flow and transport equations. Special attention is given to the highly nonlinear nature of the governing flow equation. Alternative methods for describing and modeling the hydraulic functions of unsaturated porous media are also described. "Hands-on" computer sessions will provide participants an opportunity to become familiar with the Windows-based RETC, STANMOD, HYDRUS-1D, 2D, and 3D software packages. HYDRUS-3D will be presented and demonstrated for the first time.For more details and registration visit

http://www.geo.uu.nl/hydrogeology/hydrus/scope.htm

YANG, 06/15/2005
thank you for your help.
by the way have you HYDRUS courses online ?

Jirka  06/15/2005
You can download tutorials for both HYDRUS-1D and 2D at this site and follow those.
Jirka

YANG 06/15/2005
thank you Jirka


**ID = 400, Solute unit**

David, 06/16/2005
Hello,
I would like to know the units of some data on solute. I have got some problems to have a good mass balance by adding all the different solute fluxes.
I think initial concentration is on mgN/cm3 of water and not in mgN/cm3 of soil (if I use cm as length unit and mgN as mass unit). In the postprocessing, all the tables and the graphs concerning solute fluxes are on mgN/cm of soil or mgN/cm of water. In this case, if these values units are mgN/cm of water, how it is possible to have the mass of solute leached for example and mass of nitrate uptaken.
Thank you for your answers.


Jirka, 06/16/2005
Concentration Units:
Concentration units, c, in Hydrus are [M] of solute per volume [L3] of water [M/L3], i.e., concentration in water. In general, concentration units should be consistent with the length units [L] used in the rest of the project. Then the derived variables are as follows for 1D, 2D, and 3D problems, respectively:

Transport domain – A: [L], [L2], [L3] for one-, two-, and three-dimensional problem.
Boundary length – L: [-],[L],[L2]
Water content – theta: [L3_Water/L3_soil]
Amount of solute in the soil domain (c*theta*A): [M/L2], [M/L], [M]
Water flux – q [L/T]
Solute flux – (qc): [M/L2/T]
Solute flux across boundary – (gcL): [M/L2/T], [M/L/T], [M/T]
Cumulative solute flux (qct): [M/L2]
Cumulative solute flux over boundary of length L (qctL): [M/L2], [M/L], [M]
and so on. M can be anything, i.e., mg, g, mol, meq, etc.

If you, however, look at the governing convection-dispersion equation, you can see that the concentration term appears in each term. Consequently, the concentration [M/L3] does not necessarily have to be given in the same length units [L], as the transport domain. However, one then needs to convert variables derived from concentration (given above) considering this mismatch of units (between concentration and length units).

Jirka

**ID = 401, Solute transport through vadose zone**

Jason, 06/17/2005
Hello,
I want to simulate chromium transport vertically through 5 meters of silt loam in the vadose zone but don't seem to be having much success. More specifically, I want to know what the groundwater concentration will be at the bottom boundary (water table) after a certain amount of time so I can take it and simulate horizontal flow to a well in the saturated zone. Can someone help in setting this up? I'm assuming constant water and solute flux at the top (ground level) but not sure what the bottom should be. When setting initial concentration, do I select the entire region, or just the top? When I do set this up to run correctly, where do I read the final concentration? Thanks.

Jason


YANG, 06/17/2005
Hello Jason,
When you set this up to run correctly,you can read the final concentration at this any time by the button in this picture.
maybe it is useful for you.
YANG


Valliyappant, 06/18/2005
Hi Jason,
Answer 1:
You can use free drainage BC if the groundwater level is very deep and thus does not affect flow in the transport domain.
OR
You can use deep drainage BC which relates flow at the bottom of the soil profile to the position of the groundwater (that groundwater must be above the bottom of the soil profile, i.e., within the transport domain).

You can read the final concentration from the post processing side.
1. choose solute flux dialog box
2. then choose which ever boundary you have used.
OR
If have you have observation nodes then choose observation nodes dialog box.
Answer 2:
You need to select the whole domain for the initial pressure distribution. But not for setting the BC's.

Hope this helps.

**ID = 402, hT.out and h.out?**

YANG, 06/20/2005
Hello,
Everyone,I am a new user, I wonder,
1. Is it possible to change h.out to hT.out for hydrus2d version 2.101 in order to display the Hydraulic total?
2. The value of initial condition(I use prssure head)could be only obtain by test direct or according to the curve water content?


Jirka, 06/20/2005
1. HYDRUS-2D displays pressure heads from the file h.out. Thus if you rename hT.out as h.out, it will displays your total head values.
2. Not clear.
J.


YANG, 06/20/2005
Hello Jirka,
I changed it , but I cant open the Grafical Display of results. am I wrong ?


Jirka, - 06/20/2005
Check that both files have exactly the same size. If they do, then it should work. If they do not, then it should not.
hT.out is not a regular feature of HYDRUS-2D.
J.


**ID = 403, Error**

Jason, 06/20/2005
Hello,
When I go to run a calculation I keep getting the error message: "Error in FE mesh. Check that you do not have individual nodes on the boundaries". Any thoughts? When creating my mesh, I've been told to choose fundamental. What's the differnce between "fundamental" and "make mesh" options?
Also, is it OK just to use "retangular" geometry if indeed my area is rectangular, or should I use "regular" for a more precise simulation. Thanks.
Jason


YANG 06/20/2005
VIDEO TUTORIALS
http://www.pc-progress.cz/Fr_Hydrus.htm
it is very good.
sorry,
the location of Video tutorials is
http://www.pc-progress.com/Fr_Products_Meshgen.htm

Jirka 06/20/2005

Jason,

You can not have individual nodes on the boundary. Boundary can be composed only from lines, splines, arcs or semiarcs. Individual nodes can be inserted only in the transport domain. Usual mistake is that users place individual nodes in the corners of the transport domain and then connect them with lines. That can not be done in this way.

If you have rectangular region, you can use both internal (simpler) or external (meshgen, more complex) mesh generators.

Jirka

Mirek, 06/20/2005

The Video tutorials for Meshgen are at

http://www.pc-progress.com/Fr_Products_Meshgen.htm

Mirek

## ID = 404, Vertical boundary conditions

Ena 06/23/2005

Dear all,

I am interested to see the changes in pressure head with depth in unsaturated and saturated zone due to infiltration (rainfall) and drainage (drying in the unsaturated zone when there is no rainfall).

At this stage, I am using a simple rectangular domain. I assigned the initial condition for the pressure head where it also shows the position of the initial water table. For the boundary condition, I applied atmospheric boundary at the surface and no flow at the bottom of the boundary.

My problem is to find the best vertical boundary condition. I have read the definition in the manual and I first thought that I already understand what each term mean.

I have tried for different vertical boundaries (i.e. no flow, constant pressure, variable pressure and variable flux) in order to see how these boundaries affect the results. For this case, the area of interest is at the middle section of the domain. I plotted a graph of the pressure head vs. depth for different vertical boundaries and for each time step to see the differences.

At the beginning, any applied boundaries produced the same results and changes of the results started to occur at the certain amount of rainfall and time – which I think is acceptable.

However, my concern is why variable pressure behaves and produced the same results as constant pressure, as if the pressure head at that boundary remain constant just like when applying constant pressure? I thought variable pressure allows variation of the

pressure head at the boundaries? What is the different between these two boundaries then?

Another question is why does variable flux produced the same results as no flow boundary? Is that right? What does it make them behave this way?

Any advice will be much appreciated

Ena


valliyappant 06/24/2005

Hi Ena,

If you don't mind can you explain what do you exactly mean by vertical boundaries?

Is it the boundaries at the sides of your domain (left and right hand side)?

If so, what is the purpose for your vertical boundaries?

1. Are these boundaries acting as a inflow/source (for water flow) to your domian of interest?

(OR)

2. Are these boundaries acting as a outflow(for water flow) from your domian of interest?

(OR)

3. Are these boundaries acting as a gradient for flow from the left/right to right/left in your domian of interest?

(OR)

4. Are these boundaries acting as a regional flow?


Ena, 06/24/2005

Hi Valliyappant and all,

Yes, the vertical boundaries are the boundaries for the left and right hand side of the rectangular domain – sorry for not explaining it clearly.

At this stage I am assuming that the boundaries are acting as outflow from the domain of interest. I do understand the outflow won't be happened at all if I put no flow boundaries at both sides.

Actually I am more concern about the differences between the variable pressure and constant pressure boundaries. When is the best scenario to use them? My previous question is actually regarding the pressure head changes with depth at the end of rainfall (infiltration). For two analyses with the same input parameters but the only difference was - in one analysis I applied variable pressure boundaries at the left and right hand sides of the domain and the other analysis I applied constant pressure boundaries also on both sides (with the same value of constant pressure at both sides). At the end of the infiltration analysis, the results are exactly similar for both analyses.

I am just wondering why variable pressure boundaries at the left and right hand side of the domain doesn't allow any increase in water table and hence, doesn't allow changes in pressure head at those boundaries at all. From the graphical display results, it seems that the variable pressure boundaries controlling the movement of water table from going up at the right and left hand side of the domain exactly how constant pressure boundaries behave.

Thank you

Ena

Jirka 06/24/2005
Ena,
What values of variably pressure head have you specified at vertical boundaries? How did you change these values with time? Do you realize that you actually have to specify some values for this type of BC?
J.


valliyappant 06/25/2005
Dear Jirka,
How can i specify values for variable head BC?
I didn't realize that I could specify values for variable head BC. I just opened an hydrus2d project and tried using variable head BC, hydrus2d didn't ask me for any values.

I thought that, since it is variable head BC hydrus2d will take values for this BC based on the initial hydrostatic pressure head and the change in pressure at each time steps due to change in the water balance within the domain.

Can you please explain how variable head BC works?


mizo316, 06/25/2005
When you select variable time boundary, a new input step is added (just after the soil hydraulic parameters). You would have to insert the starting and final time and the value of head or flux for each step.
An example on variable time head boundary:
t = 0.5 days, H = 5m
t = 5 days, H = 2m
This mean that from time = 0 days to 0.5 days, head = 5m
From time = 0.5 days to 5 days, head = 2 m.

You can check FAQ 29 for more details.
Hope that helps.
Mazen


valliyappant 06/25/2005
Dear Mazen and Jirka,
Thanks for your reply Mazen. I didn't realize this option/facility till date.
Have you used this option in any practical case studies? OR Have you applied this variable head BC in any of your work?
If so can you give me an example in words for the following please?
1. Groundwater level [L].
2. Mainly an example for other applications of time-dependent prescribed head boundary condition.

Thanks for your time and help.

Ena 06/26/2005
I also didn't realise that I have to put some values for the variable head BC. What I did was actually highlighting all the nodes for the left and right hand side of the rectangular geometry to have variable head BC and just run it without putting values. Say, I applied rainfall of 0.1 m/day for 25 days on the surface, the water table is initially at -10 m depth, and the initial hydrostatic pressure head at the top is -10 m and the bottom of the pressure head is 40m (linear distribution with depth)
And, if I assume that the water table increase at certain time, then,
Time (day), Precip (m), GWL(m) – assume values.
5 , 0.1, -9.75
10, 0.1, -9.5
15, 0.1, -9.25
20, 0.1, -9
25, 0.1, -8.75

I am wondering whether that is right. The way I understand it, it means that I am implying all the nodes on the left and right hand side to have the same head value (of GWL values that I applied) – Basically how Hydrus interpret these values?
Can I actually use this type of BC for my left and right sides of the rectangular domain?.

Thank you for your help
Ena


mizo316, 06/26/2005
Hey --
A type of application for this would be subsurface injection of liquids. If the rate is not constant throughout the time period, then this option becomes extremely valuable (Thanks Jirka and others). For example, you want to inject liquid at 5 m2/hr for 3 hours, turn it off, inject after a period of time at a different rate for any duration. Basically it is a transient mode (on/off).
GWL is where you would input the pressure head values if head is your variable.
rGWL is where you would input flux if flux is your variable.
Note that you insert flux values in a negative sign.
Hope that helps.
Mazen


**ID = 405, Two-site concept**

Jodie, 06/24/2005
I'm trying to use hydrus-1D to study the transport of contaminants (both solute and colloids) in the mixture of two kinds of porous media with different surface chemistry.

Question 1: Hydrus-1D 2.0 implemented two sites concept, one is instantaneous sorption site, the other is kinetic sorption site. Can I use version 2.0 to finish the solute transport in the mixed media by assuming one site sorption is instantaneous while the other one is non-equilibrium sorption?

Question 2: For the colloid transport, I'm so glad to find that the new feature in version 3.0 which could be used to do particle transport in porous media now. And for the mixture of two materials, I can use S1 and S2 (equ 3.23 of version 3.0) to represent the concentrations on two solid surfaces, can't I? But the problem comes: comparing equ 3.23 in version 3.0 to equ 3.2 in version 2.0, the good thing is that equ 3.23 in version 3.0 provides two non-equilibrium sorption on solid phase and one equilibrium sorption site. But there isn't concentration term in the gaseous phase. Does that mean it only can be used to the saturated condition? If not, which term can be used as the concentration in gaseous phase? Se by assuming instantanesou interaction on air-water interface? But how can I incorporate the air content to equ 3.23? I'm very confused about the equation. If I have some understanding mistake, please don't hesistate to indicate. Thank you very much.

Jodie

Jirka, 06/27/2005
Jodie,
Question 1: Both versions of HYDRUS-1D allow you to use two-site sorption concept with one fraction of sorption sites being at equilibrium with the solution, while sorption on the second fraction of sorption sites is assumed to be kinetic.

Question 2: When you use two kinetic sorption sites model (i.e., eq. 3.23), which is intended for simulating transport of colloids (viruses), then transport in the gaseous phase is disabled. I do not think that one can use Henry's law to describe distribution of colloids between the liquid and gaseous phase (these sobustances are not volatile). The model can be use for transient variably-saturated conditions (note that water content can be time-dependent). If you want to consider sorption to air-water interface, then you need to use of the kinetic sorption sites for that, and recalculate the attachment coefficient for that purpase (considering instead of the bulk density, surface area of the air-water interface).

Jirka

## ID = 406, Random field

mizo316, 06/24/2005
Hey Jirka --
I have asked you this question before, but I would like to confirm my understanding for it. When a random field is generated for a given domain, the hydraulic conductivity of the material defined in the soil hydraulic parameters table becomes the reference k and a scaling procedure will create the realization. After generating this realization, is the geometric average k qual to the reference k, or are they different?

Thanks -
Mazen

Jirka 06/27/2005
Ks specified with soil hydraulic parameters should be the mean value. Scaling Parameters describe just the statistics of the distribution.
J.


mizo316, 06/28/2005
Jirka --
I took the generated scaling values in the Domain.in file and calculated the geometric mean for those values. The Geometric mean was not equal to 1. So that means that ks specified in the soil hydraulic parameters cannot be the mean value of this realization. At this point, I am dougbting if the generated values follow the lognormal distribution selected in the input.
I appreciate your feedback on this subject.
Thanks --
Mazen


## ID = 407, File recovery

mizo316, 06/26/2005
Hi Jirka --
is there a way to recover a deleted file (*.2D)? I have the output folder of that file and just the *.2D file is missing.
Please let me know and thanks -
Mazen


Valliyappant, 06/27/2005
Hi Mazen,
You can recover a deleted *.h2d file by using any *.h2d file and just renaming them using the names of particular projects (folders). The *.h2d file is mainly to inform HYDRUS-2D about an existence of the project. So you haven't lost or missed any file.


YANG, 06/27/2005
you can look this link, Hydrus-2D frequently asked questions
the 14th question "Missing *.h2d files"
http://www.pc-progress.cz/Fr_Services_Hydrus_FAQ.htm


## ID = 408, Anisotropic, homogeneous domain

YANG 06/27/2005
Hello everyone,
I am a new user.
I have read the tutorial and the help of HYDRUS-2D.but I
didn't find an example about how to specify an anisotropy.
so I wonder if I simulate an anisotropic,homogeneous domain,

for example Kv(conductivity vertical),Kh(conductivity horizontal)
Kv=2Kh, Angle w=0,
so I specified
The first principal component K1A=1
The second principal component K2A=2
I am not sure whether is right or not?
thank you !


valliyappant 06/27/2005
Hi Yang,
According to the value you have given, vertical conductivity is twice that of horizontal conductivity in your domain.


## ID = 409, Domain vs domain.in

mizo316 06/29/2005
I have sevreal domain files that have an in extension. I cannot open those files. Any reason for that?
Thanks -
Mazen


valliyappant 06/29/2005
Hi Mazen,
The file named Domain with any extension are binary files. You can convert them into text files and read the information. For more information please refer the chapter 'input data' in hydrus2d manual.
Hope this helps.


mizo316, 07/13/2005
Hi -
when i am opening the domain file, i see weird symbols and scripts. This apply to some files and not all files. I suppose the difference is havain a domain.in and a domain file. Why there is 2 types of files? Is it because some were simulated using a different version of Hydrus? anyhow, please let me know how to properly read the doamin files (domain.in).

Thanks -
Mazen


Jirka, 07/13/2005
Mazen,
Domain properties are stored by default in a binary file Domain.in. It is much faster for different module to communicate when they read or write binary files than ascii files. There is, however, option to save this information as an ascii file (Command File->Save as ASCII in the Boundary module). Then instead of a binary file

Domain.in, Hydrus creates an ascii file DOMAIN.dat. The only reason for that (i.e., to use ascii files) is when you want to manually edit it. Otherwise I do not see reason for using ascii files.

Jirka

## ID = 410, Mass balance check: balance.out vs T-level.out

John, 06/30/2005
Gentlemen,
I am using HYDRUS-1D (thanks for the improvements!) to estimate daily net infiltration rates (either + or -) at ground surface using daily precip and PE rates via the atmospheric boundary condition (0 surface layer and precip rates are too low to generate runoff). The results are to be used as inputs to a 3D variably-saturated model that does not offer an atmospheric boundary condition, but will use the same 60 year sequence of daily precip and PE values.

My first approach was to use the sum(vTop), sum(vBot), and Storage values, as presented in the T-Level.out file, to estimate a change in vTop, vBot, and Storage for each day. That is, I assumed that the change in Storage between day i and day i-1 = vTop(i) + vBot(i), all of which can be calculated from the T-Level.out rows corresponding to days i and i-1. However, I found that the discrepancies between the change in Storage and the incremnetal fluxes through the top and bottom boundaries between day i and i-1 are often 1-4% and can reach 12-38%, depending on the hydraulic properties used. The Balance.out WatBalR entries for those days are routinely < 0.01% and I believe spatial and temporal discretization are more than adequate. Why do the change in storage and change in influx+outflux, calculated for each day, differ by so much more (0.5-38%) than the WatBalR values (<0.01%) for the same days?

Another approach is to use the top flux entry from a Balance.out report for each day. However, I noticed the top flux for day i in the Balance.out is usually significantly less than the difference in sum(vTop) for days i and i-1 except after several continuous days of no rain (only evap). The disparities are greatest for rainy days and the drying days immediately following the precip events.

I am puzzled by these disparities in water mass fluxes between the T-Level.out and the Balance.out files. I shall be very grateful for your advice about which, if any, of these estimates I should use to specify net infiltration at the surface.

Thanks and regards,
John Sigda, Ph.D.

Jirka 07/03/2005
John,
WatBalT and WatBalR are calculated from the sum(vTop), sum(vBot), and sum(vRoot) internally in the program. Thus I do not see how there could be large

differences in values you calculated and those reported. Except that these values are printed into t_Level.out only with a certain precision (only several valid numbers, not full float precision), and WatBal values are for the entire calculation period. Note that vTop or vBot values are not the average values for some time period, but actual values at a particular time. There can obviously be different values recorded during a particular time interval than at the end of that interval. This is obviously taken into account internally by the code.

Jirka

### ID = 411, Program stopping too soon

elduderino3   07/01/2005
I am trying run a .5 day simulation and the time only runs to .05 day in the output. For a half day run, is it better to use min's or hrs.? I have tried decreasing my min dt, but that hasn't worked
thanks

### ID = 412, Soil parameter database

Marek   07/03/2005 :
The data base for soil parameters of silty clay and for clay lists the value of vanGenuchten parameter "n" 1.09. This means that the value of m where m=1-1/n is a very small number. If you consider a common vanGenuchten expression you would see that the relative permeability even for a saturation of 0.5 is practically 0. Consequently there could be only a very little flow rate through such soils.
Other earlier references (one of Jack Parker's publication) list n for silty clay 2.8. Such n value makes water moving much faster, and more realistically in my opinion.
I would appreciate any comment on this discrepancy. Thank you.
Cheers,
Marek

Jirka 07/03/2005 :
Soil hydraulic parameters for the soil catalog that is implemented in HYDRUS comes from the paper of Carsel and Parish, WRR (I believe around 1988, I do not have it here at home) and it was also reported in the manual of the RETC code. These numbers are average numbers for various textural classes (in the USDA textural triangle) and were obtained by analyzing large datasets. Obviously, we can not change these numbers. We do, however, recommend using the option with the air-entry value of -2 cm for soil having the n parameter smaller than about 1.2. This should significantly increase the relative hydraulic conductivity.
Optionally, you can use values from the Rosetta module that does (because it was obtained on different databases) provide slightly different numbers that Carsel and Parish.

Jirka

marek 07/04/2005
Thank you Jirka,
I also found the same reference that is used for the HYDRUS database and it is indeed Carsel and Parish, 1988. It is also consistent with another source by Rawles, Brakenisiek and Saxton, 1982, who gave only slightly higher n values of 1.127 and 1.131 for silty clay and clay, respectively. Actually, this would be good news for my client who claims that his site could not contribute to the problem, but it does not go very well with the reality check because groundwater is contaminated anyway.
Cheers,
Marek


**ID = 413, Infiltration with variable bc**

Kushner, 07/06/2005
Hello,
I am running a basic infiltration model with time in seconds and length in meters. Everything seems to run well, except in the output files I get pressure heads in the -8 to -9 meters. I feel this is way too low, should be more like -.5 to -1.5 m. As far as I can tell, all numbers have correct units. My BC's in the upper are a "variable flux" and the lower is "constant pressure head." Any suggestions?

Also I am not able to locate what the variables of "AXZ, BXZ, DXZ, MAT & LAY" mean in the "soil profile summary", I am guessing they have to do with solute and heat transport, but I can't figure it out.
any suggestions or comments are appreciated


Jirka 09/08/2005
AXZ, BXZ, DXZ are scaling factors with respect to pressure head, hydraulic conductivity and water content.
MAT - material number (for heterogeneity)
LAY - subregion number (for mass balance calculations)

J.


**ID = 414, MeshGen 2D**

EfeT, 07/07/2005
Hello,
I am a new user and currently applying this software in final thesis ,i could not open window for Geometry and Finite Element(FEM) Editor under the General Geometry Type,that the program has performed illegal operation.
Has anybody encountered this problem before?Please tell me what to do.
Thanks

EFETOBORE E

Mirek, 07/07/2005

Hi EFETOBORE,

this is really strange because the Meshgen has been used for more than 8 years by many users without any problems. First of all I'd recommend you to check if it is installed correctly. Try to re-install if (the latest version is at http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm) or install it on another computer. Then try to open your project. If it fails, please send me the project. Does it fail always when you try to open the Meshgen window or only in a specific project? What Windows do you have?

Mirek


EfeT, 07/11/2005

Hi Mirek,

It fails always when i open the Meshgen window.The type is: Hydrus-2D for Win NT and Win 95,version 2.0,and i learnt that my school purchased it in the year 2000,do we need to upgrade?

Thanks

Efetobore,E


Mirek, 07/11/2005

Hello Efetobore:

Download the latest version (http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm) and try if it works on the computer where you have problems now.

A/ If it works then I'd recommend you to upgrade. I think it should be for free if you have version 2.0. Contact IGWMC (IGWMC@mines.edu) to get more information and the license.

B/ If it doesn't work then probably there is a DLL-problem/conflict in the Windows system directory (the Windows system can be damaged). Finding such problem can be very difficult and time-consuming. Try to install Hydrus2D on another computer if you can. If you can not do this then we will continue later.

Mirek


## ID = 415, Problems with convergence of water flow

Mvcallaghan, 07/08/2005

I am in need of help in figuring out why I am having so many problems with model convergence. I require very stable model runs to allow for an extreme sensitivity analysis on model input parameters (extreme, but realistic).

I am trying to test the usability of Hydrus 1D for field-scale salt transport modelling. I am having ongoing problems with model convergence of the water flow component. I have laboratory derived values for Ksat and the moisture retention curves. Typically my soil columns contain three soil types. My upper BC is Atmospheric with Surface

Runoff, and my lower BC is Free Drainage. I have tried both Pressure Head and Moisture Content initial conditions. The Moisture Content initial condition generally gives me convergence more often than the Pressure Head IC. My soil column in 5000 mm deep, with the water table at an annual static level of -2500 mm (fluctuates seasonally). I have entered daily precipitation (up to 15 mm/day) and potential evaporation values. My simulation duration is up to 10 to 50 years. I have already produced good model results from another program (a finite difference one). Now I am trying to produce results in Hydrus to compare to it.

The main instigators for the non-convergence seem to be the low Ksat values for my soils, ranging from 0.01 to 10 mm/day (can be highly consolidated), and from the moisture retention curves I have. Typically my theta sat. values are 0.4, and decrease to only 0.3 to 0.25 by 1500 kPa suction pressure. Basically, my soils like to hold onto water. I have tried both the Brooks-Corey and the Van Genuchten-Mualem models also with the 2cm Air Entry value, and so far the Brooks-Corey model gives me the best convergence. I have tried various node spacings as small as 10 mm; typically I use 100 mm with my finite difference model. My HCritA value is -150 000 mm (-150 m).

I have examined the NOD_INF output files for signs of unrealistic moisture contents, fluxes, or pressures, and I haven't found anything wrong. I'm sure I'm missing something very basic. I just need someone to point it out. lol. Even when I can get a successful multiyear run, the model is so unstable that if I change any parameter by 1%, I lose convergence.

Some of my ideas on what might be wrong:
1) large discontinuities in the magnitude of the Ksat value are causing instability (my soils can experience 100 to 1000 fold differences in Ksat)
2) extreme drying and wetting of the upper BC causing instability

Any ideas would be very helpful and appreciated.
Mike


Jirka, 07/09/2005
Mike
We have run thousands of HYDRUS-1D simulations for API for extremely heterogeneous conditions (both clay and sand layers) and both wet and dry climates
 (see Hendrickx, J. M. H., G. Rodriguez, R. T. Hicks, and J. Simunek, Modeling Study of Produced Water Release Scenarios, Regulatory Analysis and Scientific Affairs Department, API Publication Number 4734, 146 pp., 2005.)
for hundreds of years to predict transport of brine releases. Once we got reasonable temporal and spatial discretization, as well as iteration criteria, the model ran smoothly. Thus your problem will be in these variables and parameters.
Spatial discretization: In general no model can be run at 10 cm discretization at the soil surface and provide accurate calculations for infiltration or evaporation. At the soil surface, you discretization should decrease to about (or less than) 1 cm. Similarly at interfaces between different soil horizons, if the soil materials are dramatically different.

Temporal discretization: The model can use large time steps during period when not much is happening. However, in few periods, e.g., infiltration into dry soil, the code may need small time steps. Do allow small time step (on the order of 1 cm).

Soil hydraulic properties model: If you have clay in the profile, use VGM model with -2 cm air entry.

If you use these values, I do not see why you should have problems with your simulations.

Jirka


mvcallaghan, 07/13/2005

Jirka,

Many thanks for the quick reply. Reducing the DX at the surface to less than 10 mm resulted in much-improved model convergence, as far as wetting and drying on the upstream boundary is concerned. My minimum Dt is less than 0.1 s. The source of instability appears to have been the high negative pressure gradient at the surface during drying events. I am still having some stability problems in some of my runs, but hopefully they won't affect my progress too much.

Now that you've solved my upper BC stability issue, I have a problem with plant transpiration. During my simulations (using an upper Atmospheric BC with Surface Layer), the upper BC becomes saturated and switches to a surface layer, resulting in a shutdown of plant transpiration. If I were to use the Surface Runoff BC, then there would not be a surface layer, and the plant transpiration would therefore recover faster as the soil column would dry faster following the rain events.

However, I'm having problems using the Surface Runoff BC. My model runs generally have poor stability when using Surface Runoff, as opposed to the Surface Layer BC. This is due to a behavior I observed during a wetting cycle. Surface runoff was generated normally when the soil column became saturated. After the rainfall and runoff ceases, the surface infiltration rate vTop appears to set itself to the Ksat of the surface soil (29 mm/d). Since the soil column is saturated, the infiltration rate of the surface soil is limited to the lowest Ksat in the soil column (in this case 0.02 mm/d). The large difference in infiltration rates between the first and second nodes causes instability and the model blows up (at least that's my theory). Of course this scenario is likely only to develop in a soil column with a low conductivity soil layer, low evapotranspiration (e.g. winter snowmelt) and a shallow water table. Do you have any suggestions as to how to remedy this type of problem?

Regards,
Mike

P.S. Thanks for the tip on the API study. I had already read it. The number of model runs was truly impressive and the scenarios were quite exhaustive.

Jirka, 07/13/2005
Mike,
If the profile is saturated, then based on the model of Feddes (1978), transpiration is indeed shut down. The roots can not transpire since they lack oxygen. See the model of Feddes (1978) in the manual on p. 14.

Jirka


Preecha 09/06/2005
Hi Jirka,
I also run into some convergence problems when using the Brooks-Corey function during the periods with large precipitation.
The soil is clay with Ks of 10-7 cm/s, alpha = 0.052 cm-1, and n = 0.43.
I tried to refine the vertical spacing at the surface to as small as 0.6 mm, min. time step as small as 10-12 day, and tolerance for water content of 1e-6, but the convergence problems still exised.
I did not have the convergence problems with the van Genuchten function.
If you have any suggestions, I really appreciate.
Kind regards,
Preecha


Jirka 09/08/2005
When you use the BC function, disable the interpolation tables or select lower limit above air-entry value.
J.


## ID = 416, Root water uptake problem

Jun, 07/08/2005
I'm running a project with the consideration of root water uptake. I have two different plants. Their root density are 0.2,0.2,0.6 and 0.3,0.3,0.4 respectively. If I want to put these two plants together, how can I fill the "Root" column at "soil profile summary"? Can I set the distribution as 0.5,0.5,1.0?
I'll appreciate any help.


Jirka,  09/08/2005
Yes, that would be correct.
Root distribution constant are relative numbers. The code will internally normalize that.
J.

**ID = 417, Hydrus book**

Jason, 07/12/2005
Hi,
I sent an email to hydrus@optusnet.com.au to order a book but have not heard from them in a couple of weeks. Does anyone know the best way to obtain it? I couldn't find a copy on Amazon or Ebay either. Thanks.
Jason

YANG, 07/12/2005
You could left your infos in this forum, the administrateur will see it,

Valliyappant, 07/12/2005
Hi Jason,
I have got the copy of the order form sent to me by the publisher. If you want I can send it to you through e-mail as an attachment. You can fill the form and send it back to the publisher.
Fax the form back to the publisher will be better option. It is fast and you can get the book in less then three weeks time if you are in urgent need.
PLEASE NOTE: If you haven't heard from the publisher then it is better to wait for their reply. Because they may not have copies or there should be some reason for no reply. I did get reply from the publisher.

Jason 07/12/2005
Thanks Valliyappant. My email address is: jdm24@unh.edu. Do you know how long it usually takes to obtain it? I'm getting the sense that this book is critical to HYDRUS-2D success.
Jason

Jason, 07/12/2005
Ignore that last question. I didn't see the rest of your posting :)

Valliyappant, 07/12/2005
Hi Jason,
Had sent you an e-mail with the order form attached.

**ID = 418, How could I see the border of the vadose zone?**

YANG, 07/13/2005
Hello,
Everyone,Hello,Valliyappant :)
Do you know if want to see the border of Vadose Zone,How

could I do in the Hdyrus? because I only see the line equi-pressur head line in Hdyrus,but if I want to see the equipotential line, how could I do? t.out,th.out?there isnt another methode?

ps: I suppose that the equipotential surface correspond the border of Vadose Zone,

## ID = 419, Courses

Jason, 07/13/2005
Does anyone know of any upcoming HYDRUS-2D courses in the US?

Jirka, 07/13/2005
May 25 - 26, 2006 HYDRUS course
International Ground Water Modeling Center (IGWMC), Colorado School of Mines, Golden, CO
Look at the news section of our site.
Jirka

## ID = 420, Surface runoff

Uwe, 07/15/2005
Hi,
there was some discussions about runoff in the forum, and I am not sure if I quite understand things well. I thought that hydrus2d doesnt evaluate surface runoff. After looking once again into the manual, I think its possible to calculate, when evaporation = null.
is this ok:
runoff = vAtm minus rAtm from v_Mean.out

regards uwe

Uwe, 07/15/2005
I use version 2.007 - the older v_mean.out!

Jirka, 07/15/2005
Hydrus evaluates runoff as an difference between precipitation (vTop) and infiltration (rTop). Infiltration is different from precipitation only after ponding at the soil surface is reach. New version of Hydrus writes runoff into the v_mean file (one of the last column), but it does not display it graphically.

Jirka

Uwe, 07/15/2005
< difference between precipitation (vTop) and infiltration (rTop).
"vTop" and "rTop": thats "vAtm" and "rAtm" in v_mean.out?

uwe, 07/15/2005
Right.
 (vTop) and (rTop) are in HYDRUS-1D
"vAtm" and "rAtm" are in Hydrus-2D

## ID = 421, Drip irrigation and HYDRUS 2D

Rosy, 07/15/2005
Hallo
I'm italian student.
Can you help me?
I must reproduced the simulation of Simunek. This simulation is in article "Comparison of HYDRUS 2D simulations of drip irrigation with experimental observations" published in August 2004 by Journal of Irrigation and drainage engineering. Vol 130 No. 4.
I try to do it but it is unsuccessfull!
Please help me
Thanks for your time and help
Rosy

Jirka, 07/15/2005
Rosy,
I will forward your email to Todd Skaggs and ask him to send you one of the HYDRUS projects carried out for this paper.

Jirka

## ID = 422, Fitting data

Sharendr, 07/19/2005
Hi!
These are my points from laboratory data. I tried to fit these data by inverse method in deterministic non equilbrium model. I found difficulty in fitting . Could one of you help me to fit these data.

Once I posted these data they became once. But I think you can identify that in each row there are 2 data (hour and concentration- concentration begins with 0)

Hours Concentration ( g/l).
0.5 0.18965
3.5 0.191254

8.5 0.196524
14 0.206616
21.5 0.212919
69.5 0.215351
117.5 0.216162
189.5 0.217243
261.5 0.219405
309.5 0.218054
357.5 0.211838
405.5 0.207243
453.5 0.207243
543.5 0.197243
591.5 0.197243
640 0.17542

Please help me to fit these data.
Thank you.
harendra


## ID = 423, Fitting

Sharendr, 07/19/2005 Hi!
When I fit the datas, I couldn't fit (0,0) point as one of my observations? How can I fit that point? The program didn't allow .
thank you
harendra


Jirka, 07/19/2005
harendra
I do not know if I understand correctly and which program within the STANMOD family of programs you are using. Does (0.0) mean zero time and zero spatial coordinate. Obviously it does not make much sense to fit concentration at time zero, since that does not depend on any parameters. Time zero is given as the initial condition.
J.


Sharendr, 07/20/2005
Hi!
The (0,0) indicates zero time and zero concentration. In breakthrough curves that point is the initial point in effluent. So how can I include that point?

THANK YOU
harendra

ntoride, 07/20/2005
harendra:
CXTFIT2 originally allowed to have (time ,conc) = (0, 0) at a certain depth at the beginning of BTC data for users like you although it does not help anything to estimate parameters. As Jirka mentioned, using data at t= 0 does not make any sense to estimate parameters because the initial condition is the condition you need to specify as similar to the boundary condition. The CDE will be solved for the initial and boundary conditions. Because of this reason, the STNAMOD interface dose not allow (0,0) in the observed data.

Nobuo


## ID = 424, H2D heat transport

Darren, 07/20/2005
Hello,
I have a question about the heat transport boundary conditions. After inserting the bc, what does "Pointer to the direction of boundary conditions" designate, and why does it only ask that after inserting the first heat bc? Also, how do I specify whether the bc is a time variable condition? Thanks for your time.
Darren


Jirka 07/20/2005
Darren,
"Pointer to the vector of Boundary conditions" selects value from "Heat Transport Parameters" dialog. If the pointer is 1, it will take value from TBound1, if the pointer is 2, it will take value from TBound2, etc. You should get this whenever you want to specify temperature boundary condition on boundary that is designated as time-invariant (which is based on water flow BCs). Time variable BCs are atmospheric, and variable head and flux BCs, others are time-invariant. For these you need to specify temperature BC in the TIme-variable BC dialog.
Jirka


## ID = 425, Simple solute flow

Jason,07/25/2005
Hi All,
I didn't want it to come down to this but I'm having serious issues with what seems like a simple problem.

All I want to do is simulate metal transport from a source at the ground surface to a well down gradient under steady state conditions (i.e. leaching rate and precip). This seems as if it's going to need two simulations: 1) vertical flow through the unsaturated zone to the water table and 2) horizontal flow to the well.

I'm using rectangular geometry b/c there is not variation in topography or head; no "time varible boundary conditions" because of steady state conditions.

I guess my questions are:

1. What boundary conditions would be best suitable for this problem? Constant head or constant flux? The water table is shallow (3-5 m)and remains fixed throughout the course of the transport. I am especially confused as to what should be used to represent the well;

2. How does HYDRUS differentiate between saturated and unsaturated conditions? Do I need to play around with water content?

I appologize for all this. I'm just a confused grad student.
Jason


Jirka,07/25/2005
Jason,\
If you have a point source at the surface and then groundwater flow to a well, then it is not a two-dimensional (nor axisymmetrical three-dimensional) problem and you can not solve it with Hydrus-2D. You need to either use some other 3D program or wait for our release of HYDRUS-3D in the fall. If, however, the solute source can be assume to be axisymetrical as well as the water flow, then you should be able to use exisymetrical option of HYDRUS-2D. I would probably specify constant head at the wall of the well, which must be in the centre of the symetry.

Jirka


Jason,07/25/2005
Jirks,
Thanks for replying. All I'm going to do for now is concentrate on 1-D flow through the unsaturated zone. For this, I'm going to treat the surface as a constant flux boundary for solute and water flow. For the bottom, I'm going to use constant head and variable flux boundaries for water and solute flow respectively. Does this sound right? At the bottom boundary, should the head value be negative representating the water table depth? My goal is to get a contaminant flux at the water table so that I may take that flux and use it in another model. Thanks.
Jason


Jirka,07/25/2005
Bottom BC should be zero head for water flow (representing groundwater) and zero gradient (free drainage) for solute transport.
J.

**ID = 426, Extraction well**

mizo316,07/27/2005
Hi Jirka -
Do we have an example on extraction wells? To model something similar, should I use axi-symetrical option? what type of boundary condition to use?
Thanks -
Mazen

Jirka 07/27/2005
Mazen,
You need to use axisymetrical fromulation. You can use both head or flux on the wall of the extrcting well.
Jirka
PS: I'm taking a five weeks vacation and will not be able to answer questions until September 1, 2005.

mizo316,07/27/2005
Hey Jirka -
Basically, extraction should always be the opposite of what we use for injection?
So, I would use negative flux to extract water out?
In case I wanted to use time boundary conditions, then i would have to input the seuence of extraction in +ve flux?
Thanks and Hope you enjoy your vacation (Europe, I assume?)
Mazen

Valliyappant,07/30/2005
Hi Mazen,
That's right.
Negative flux to extract water out of the domain. Positive flux for time dependent boundary.

mizo316,08/02/2005
Hey V.
Have you tried running some exercises on extraction wells? I tried a simple horizontal plane flow and its crashing. I don't want to put lot of time in it, so if you know some tricks, pass them on :)
Thanks -
Mazen

Valliyappant,08/02/2005
Hi Mazen,
yeah. I did some work on extraction wells. But I didn't manage to get reasonable results.

I had mass balance error which was between 5 and 10%. Few other output graphs seems unreasonable. So I need to do more work (and understand) to give suggestions.

## ID = 427, Confidence intervals and SE for inverse solution

Franta, 07/28/2005

I have a question regarding the statistics (standard error and confidence intervals) of soil hydraulic parameters optimized in the inverse solution. My solution has wide confidence intervals and I am not sure about the source of variability that leads to this in an inverse solution.

Here is the description of the experiment: Two different irrigation rates were applied over time and steady drain flow for five tile drains were measured in a field scale experiment. Soil has a six-layer profile with all tile drains (10-cm diameter) placed approximately in the same depth – within the third layer from the top. I have also soil hydraulic parameters obtained from the intact cores taken that have been averaged for each layer. I am trying to use an inverse solution to optimize Ks, n, and alpha parameters for all six layers using measured tile drain flows.

Here is what I did: I have plotted all measured flows against time in Excel and fit a separate linear line for each irrigation rate to obtain two "mean flow values". I got two lines with slope nearly equal to zero. I have set up a six layer two-dimensional X and Z profile with a variable boundary at the top, tile drain (seepage boundary – 10 cm diameter) in the third layer, and free drainage at the bottom. I have used the same tile drain flow data from all five drains in inverse solution to optimize Ks, n, and alpha parameters. I have started with averaged parameters obtained from the intact cores gradually changing and optimizing parameters layer by layer to get closer to the "mean flow values" (two intercepts obtained from an Excel plot). I managed to get pretty close to the Excel intercepts by changing parameters and optimize those parameters until no further change in sums of squares occurred. Problem is that statistics for the optimized parameters do not look very good - all of the confidence intervals are pretty wide with the low values in negative numbers. The only exception is the bottom most layer where I obtained tight confidence intervals and small standard error value.

What is the source of variability within the model? Is there a way to make confidence intervals tighter also for upper five layers? Are those parameters for all other layers valid even though confidence intervals are pretty wide? Is there a better way to approach the problem? Should I perhaps calculate an average for all five tile drains for each irrigation rate and use those two averages in inverse solution? Should I optimize all three parameters for only one layer with a grid only for appropriate depth and create a final run for all layers using those values?

Thanks
Franta

Rvang, 08/04/2005

Franta:

I think you have far too many parameters. The most important parameters are likely those of the bottom layer (especially Ks) that determine how much drainage you will get (and how much deep seepage will occur between the drains). Two possibilities:

1. Keep all alpha and n values fixed at the Lab experimental values. Again, the probably have very little effect on the drainage rates. Perhaps play only with the Ks values, and then mostly Ks of the bottom layers. That should (and actually already did) give you narowwer confidence intervals. This wide bands of uncertainty merely shows that you do not have enough informaiton for a unique inverse solution.

2. Built a more complex objective function (or other inverse approach) that adds more information. Whatever you measured additionally (water contents maybe?). In a way this goes back to fixing the alpha and n parameters of the upper layers to the laboratory values.

Good Luck. --Rien van G.

### ID = 428, CXTFIT inverse problem

Jbridge, 07/28/2005
I am working on a fluorescence imaging system which captures time lapse images of colloid and solute plumes passing through quartz sand.

I am trying to use CXTFIT to model the data obtained from these images. Although the images are 2D, I am generalising to 1D in the direction of transport.

Having calculated v and D from the spatial moments of the image data, I have quite successfully modelled conservative solute (fluorescein) plumes using the direct problem (flux averaged concentration, pulse input, zero initial, zero production, concentration vs. distance output).

However, I would like to use the inverse problem to fit parameters v and D to my data for comparison with my previous estimates. I set up the model with the same conditions noted above, and with my existing values for v and D as initial estimates. My data is two column (Z,C) by 416 rows reflecting the concentration profile down the column as imaged at time t=20 min

I specify an output time of 20 min but when I run the model I find:

a) although my supplied data is plotted, no curve is fitted
b) the text output file reads:

D........ .0000 <= 0, Use positive value !
R........ .0000 <= 0, Use positive value !


WARNING ! Please check transport parameters.

Checking back, I have definitely provided positive values. Is this a bug? Or am I doing something wrong? I am a new user to CXTFIT and solute/colloid transport modelling in general.

Any advice or help much appreciated!
Best regards
Jon Bridge


Ntoride, 07/30/2005
Jon:
I will have a look at your project. Please send an example project
(entire folder) with a cxt file to ntoride@bio.mie-u.ac.jp.
Thanks,
Nobuo


Jbridge, 08/01/2005
Nobou,
Thanks for your response. You'll be pleased to know however that I've solved the problem - my original Z-C data contained lots of C=0 lines before and after the position of the plume in the Z direction. On stripping these C=0 lines out of the data, the model worked fine. I'm not sure if this issue is covered in the CXTFIT manual or the help - in retrospect it seems an obvious issue but I am inexperienced!

File sent as requested for your info.
Best wishes
Jon.


**ID = 429, Horizontal drains**

mxc289 07/28/2005
I'm having a problem with the horizontal drains lower boundary condition in Hydrus1D. I'm wanting to use the simplest case of a 'homogeneous profile, drain on top of impervious layer' but the code seems unresponsive to the entrance resistance parameter. In fact if I enter a value of zero for this parameter the code seems happy to run when presumably there should be a floating point error. Ideally I'd like to set the drain spacing to a very large value so I end up with a linear relationship between the drain discharge and the reciprocal of the entrance resistance but this doesn't work as the code doesn't seem to behave in the way suggested by the manual/help files. Any help would be much appreciated in particular in regard to what drainage function is actually being implemented by the code. Thanks.


Jirka  09/08/2005
I think that it seems that the solution does not depend on resistance because you use very small values for resistance. The way I implemented Eq. 2.60 in the code does not lead to the floating point error. In the code, to avoid floating point errors, eq. 2.60 is reformulates as

$q\_dr = h\_dr/total\_resistance,$

where total_resistance is the sum of all resistances. Then obviously when the entrance resistance into the drains is zero, it does not have any effect. Specify it equal to 100 d and you will see the effect on drainage.

Jirka

## ID = 430, New user question

LisaFultz, 08/12/2005
I am working on a project dealing with the use of swine effluent in subsurface drip irrigation laterals in northwest Oklahoma. I am looking at the movement of nutrients through the soil profile and it was suggested that I use Stanmod as a model. I have downloaded the program and looked at some of the manuals for the different models, but I am unsure which would be the most appropriate for my situation. I would appreciate any suggestions on which program if any would be most useful.

Lisa Fultz

Ntoride, 08/15/2005
Lisa:
CXTFIT is in principle based on analytical solutions for the convective-dispersive equation (CDE). Although CXTFIT includes some modified models of the CDE such as the mobile-immobile model and the
stochastic stream tube model, all of them still have some simplifications such as uniform flow and water content to derive solutions. The program is widely used to estimate transport parameters from measured solute concentrations. If you are thinking to predict nutrients movement in fields, HYDRUS would have much more flexibility for various situations (HYDRUS solves the Richards equation and the CDE numerically).

Nobuo

## ID = 431, Open file error

Kathryn 08/14/2005
Hi, I'm attempting to model a simple drain situation (deep open ditch drainage) using Hydrus2D for my honours project. Each time I go to run my model I encounter an "Open file error in file :
...hydrus2d\Direct\drain14august\Select
or"
I can't seem to find a solution to the problem...I've tried modelling the same scenario a few different times now and I keep coming across this program, and the model won't run.
I'd appreciate any suggestions to get over this hurdle!
Regards,
Kathryn

Mirek 08/18/2005
Kathryn:
I'm currently on vacation but I can look at your problem next week. Please send me your project (the xxx.h2d file and xxx directory) so that I could analyze it.
Regards Mirek


Mirek 08/25/2005
Kathryn:
I think that your problem can be in a very long path to the project:
C:\Documents and Settings\Kathryn\Desktop\hydrus2d\
Projects\drain14august\Selector.in

This path is longer than 80 characters and Hydrus calculation module probably cuts several last characters and subsequently fails to open the file "Selector.in". The latest version of calculation doesn't have this limit (80 characters) but I remember there was this problem in previous versions.

The solution is very simple - move your projects to directory with a shorter path, for example C:\Hydrus2D\Projects

I was able to open your project and run the calculation but unfortunately it gives no reasonable results. Your initial time step is equal to the final time => calculation stops immediately after first step. Also some other input parameters of the project are probably incorrect and you will have to change them to get right results.

If you have problems with the project perhaps Jirka could help you after his return from vacation (in two weeks).
Regards Mirek


**ID = 432, Velocities**

Catalina, 08/16/2005
Hello
I am working with the Hydrus 2D programme to make a simulation of a drainage system. It is a steady state simulation in a rectangular domain with 2 ditches. The boundaries are defined as: bottom and lateral boundaries are no-flux boundaries; top boundary has a recharge of 0.001m/d and it has 2 ditches with a seepage face boundary conditions. When I checked the results for vertical fluxes I assume that along the bottom boundary there should be NO vertical flux but in the file there are values; also when I checked along the top boundary expecting the value of the recharge (0.001m/d)I found different values for the vertical fluxes. I tried to run the case agaign with a more strict convergence criteria but the velocity results are more or less the same.
Is there an error in the files or I am checking the wrong files or are my assumptions wrong?

Thanks in advance
Catalina

Rvang, 08/16/2005
Catalina:
Double-check how you assigned the BC's in "Boundary Conditions Editor" , or perhaps you misinterpreted the outut in the "Water Boundary Fluxes". Look at the "Furrow" example, but with a modified soil surface BC. --Rien van G.


Catalina, 08/17/2005
Hi again
I still have the problem. I am checking the file V.txt, for the nodes at the boundaries but in the no-flux boundary nodes for the bottom boundary there are values for the z direction flow and it should be zero. It also appears in the furrow example.
Catalina


Catalina, 08/17/2005
I think I know what is happening. It seems to me, that HYDRUS calculates the velocities by element and latter it gives to every node an average value of the velocity from the elements surrounding the node. It doesn't take in to account if the node is a boundary node.
Is it possible that I can get the velocity field in the center of the elements wich seems to be more accurate?


Mirek, 08/18/2005
Catalina:
I think that Hydrus calculates results (velocity vectors) in FE-mesh nodes but need to check the source code. Unfortunately I'm currently on vacation and I'll not be able to do that sooner than next week. I believe that Jirka could answer this question immediately but he is on vacation too. We will have a look at your problem next week.
Regards Mirek


Jirka, 08/24/2005
Catalina,
HYDRUS calculates the pressure head gradient in all elements surrounding a particular node, then calculate mean gradient for the node, and multiplies it with the hydraulic conductivity to get the flux. It does this for all nodes, including the boundary nodes. These calculations are, however, only for postprocessing and do not affect the actual calculations of water flow.
For fluxes at the boundaries, check the file Boundary.out, into which we print fluxes that go directly from the system of governing equations and thus do reflect the actual solution.


Jirka08/29/2005
Hello
Again me. I would like to know if the fluxes values that hydrus calculate (V.txt) are used for the solute transport?.

Jirka, 09/08/2005
Yes. J.


**ID = 433, Set slope of tank model**

Srilert, 08/19/2005
Dear Jirka,
I have problem how should i do ? when i want to set slope of tank model(hydraulic gradient). I mean i create tank model (unsaturated-saturated model) for studying movement of heavy metal when it leak from the containment from topsoil to groundwater and i want to set groundwater flow have a slope 0.001. How can i set in this sloope of groundwater flow(hydraulic gradient) in this model? This set up model similar as example 7 in manual but in this example has constant water level but in my set up model i want to change it have slope of groundwater flow (hydraulic gradient)
I 'm new users pls. advice me.
And I have another 1
question that is Kd is the same value in unsaturated and saturated zone ? I 'm not sure in theory .
And any suggestion, you can advice me.
Regards,
Srilert


Jirka, 09/08/2005
Lert,
In the boundary module you need to specify initial condition (pressure heads) at a slope of 0.001. Then you need to fix the pressure head (constant head boundary condition) at both sides.
Kd should be the same in the unsaturated and saturated zone (I think). This is because the wetted surface area is the same in the unsaturated and saturated zone (there is always some small film of water at soil surface).
J.


rvang 09/08/2005
Yes, the Kd (equilibrium Kd) is independent of water content (saturated, unsaturated, ...).--Rien.


**ID = 434, Tracer transport**

Tlange, 08/24/2005
Hello,
Subject is:
-----------
Roughly estimating tracer transport through a thick unsat zone (actually Tritium). At the end it would be nice to apply a whole Tritium input function.

Current steps:
--------------

# 1st step:
"Semi-"saturation of the profile until stability according to the infiltration rate.

# 2nd step:
Applying one tracer impulse to the profile.

# 3rd step:
Applying severeal tracer pulses to the profile - the input function.


Problem:
--------
If I have only one tracer impuls the run is ok. But then trying 2 impulses the before stable saturation gets unstable (drops continiously).
The fault is propably on my side (I just started with Hydrus1D). How to I properly define a succession of my tracer pulses? I suspect I somehow misunderstand the idea of how to define these things in the "Time Information-" and "Variable Boundary Conditions-" boxes.
Additional question: Is it possible to define or simulate a decay rate?
Thank you for help, Torsten


Jirka 08/24/2005 :
Torsten,
From your question, it is difficult to say what went wrong. Zip the project and send it to me by email (after September 1 when I return to Riverside).
Jirka


Tlange, 08/25/2005
Hello Jirka,
thank you very much for your offer. With the definition of the time boundary I made one step forward. But I will appreciate your help, though I try to solve it before September 1st ;)

Anyway I make it more specific to everybody:
-------------------------------------------------
By using these parameters below the water flow is calculated fine, but the concentration of my tracer remains zero at any observation point at any time.
Does someone sees the mistake?

# Geometry definition:
2750 m (with inclination)
Looks a bit wired, it is an eroded anticline structure...

# Time discretization:
365000 days, 10 time-variable boundary records

# Hydraulic model:
single porosity, van Genuchten - Mualem, no hysteresis

# Water flow parameters:
xyz

# Water flow BC:
UBC - Variable pressure head/flux
LBC - Constant pressure head
Initial condition - water content

# Solute transport:
Crank-Nicholson | Galerkin FE | stab. criterion = 2 |
use tortuosity factor | 1 solute | pulse duration 36500(0)

(Pulse duration looses its meaning when using time
variable boundary records? I changed it with now effects)

# Soil transport parameters:
xyz

# Reaction...
no reaction

# Solute transport BC:
UBC - concentration BC
LBC - zero gradient

# Time variable BC (just a stupid test run)
time fluxTop hCritA hTop KodTop cTop cBot
36500 -0.0005 0 0 -1 500 0
73000 -0.0005 0 0 -1 0 0
109500 -0.0005 0 0 -1 0 0
146000 -0.0005 0 0 -1 0 0
182500 -0.0005 0 0 -1 0 0
219000 -0.0005 0 0 -1 0 0
255500 -0.0005 0 0 -1 0 0
292000 -0.0005 0 0 -1 0 0
328500 -0.0005 0 0 -1 0 0
365000 -0.0005 0 0 -1 0 0

**ID = 435, In filtration**

Sharendr, 08/25/2005
Hi !
My friend is conducting an experiment in packed sand column. He is passing solute
free water through column and collecting how much sand particles present in effluent
from that column. Is it possible for him to use stanmod package? If not which package
is good for him? Will hydrus help him?

harendra

### ID = 436, Contaminated column

Sharendr, 08/25/2005
Hi!
I am packing contaminated sand ( tri chloro ethylene )in column. And I am flushing surfactant solution through that column in order to determine how much contaminant released from that column. Can I use STANMOD to model how much contaminant released from that column?

harendra


### ID = 437, Step input

Sharendr, 08/25/2005
Hi!
In stanmod package,for the step input, why we don't have time slot to enter the time value (applied time period)? Do the step input means continous input?
harendra


ntoride, 08/27/2005
As I always mentioned to you, please look at the manual regarding the
boundary value problem. Same as true for your previous questions. Nobuo
Several different functions can be used for the input concentration as a
function of time in the boundary value problem (BVP) (MODB in Table 6.7
of the CXTFIT manual): (0) Solute free water input; (1) Dirac Delta
input; (2) Step input; (3) Pulse input; (4) Multiple pulse input; (5)
Exponential input concentration.


### ID = 438, No access to soil hydraulic parameters

Kathryn, 08/27/2005
Hi all,
I can't access the soil hydraulic parameters in pre-processing menu of hydrus2D. Do I have to install anything in particular to access this section?
Regards,
Kathryn


Mirek,  08/27/2005
Kathryn:
soil hydraulic parameters should be always accessible (except while running calculation). What exactly happens? Is the menu item disabled or the dialog doesn't show up?
Mirek

Kathryn, 08/28/2005
In the pre-processing menu, under water flow parameters, the soil hydraulic parameters item shows up, but clicking on it doesn't do anything..
Thankyou for responding so fast,
Regards,
Kathryn


Mirek, 08/28/2005
This is really very strange, I've never heard about such problem.
1. Try if this problem appears in all projects and also try to create and test a new project. Soil hydraulic parameters should be always accessible.
2. If this problem is in all projects then your Hydrus2D installation is probably damaged and I'd recommend you to download the latest version (http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm) and reinstall Hydrus2D to a new directory (for example C:\Hydrus2D)
3. If the problem is only in one project I'd try to reinstall Hydrus2D too. I have no idea why soil hydraulic parameters could be inaccessible. If this doesn't help, send me the project once more. I can access soil hydraulic parameters your project "drain14august" without any problems.

Mirek


Jirka, 08/28/2005
Hydrus software uses two ocx objects. One is a table (such as Soil hydraulic parameters) and one is an x-y graph (such as observation nodes). None of these will work if the software is not properly installed (running installation program. Thus you can not just copy the software from one computer to next), since during installation these two objects are registered with the Windows registration database.

Best regards
Jirka



**ID = 439, Colloids transport**

Cobaltblue, 08/29/2005
hello:
I want to use HYDRUS1D 3.0 to estimate colloid transport parameters such as attachment and detachment coefficients .
Only one kinetic sorption site was used and no equilibrium site was considered.
When the solute transport parameter iPsil was set to 1 and SMax1 was set to a value that greater than 0, for instance, 0.5, the program was terminated with information ¡° Does not converge in the solute transport module¡± . If SMax1 equal to 0, then the program can run successfully.
SMax1 should not be equal to 0 according to equation 3.25, i want
to know why this happen.

Jirka, 09/08/2005
Parameter iPsi makes the convection-dispersion equation nonlinear. You then need to specify iteration criteria in the Solute Transport dialog window. See example Test4.
J.

## ID = 440, Solute transport boundary conditions

tom 08/31/2005
Hi,
I want to simulate the transport of heavy metals through a two layer profile. I characterized the layers with different Kd and beta values coming from my Freundlich fits.
I know the concentration of the infiltrating water (upper boundary: const. flux)which I
(1) quantified in column cBnd1 using any concentration unit I would like to see again in the results, right?
(2) How can I set the concentration in the two different layers? Initial condition? what concentration unit? I know the total HM content as well as the soluble amount(via Kd or via real measurements from lab )
Thank you in advance
Tom

PS: where can I learn about the meaning of the cBnd1-4?

Valliyappant, 08/31/2005
Hi tom,
You can learn the meaning (or what it stands for?) of cBnd1-4 from the help menu in hydrus2d software or from the manual.
You can set the initial solute concentration using
 (1) Boundary conditions editor
(2) Condition menu
(3) Initial condition.
Note: You need to split the region when you select and input the initial solute concentration.
The concentration unit is mass/mass (M/M). I have came accross mg/kg of sediment. You need to maintain unit consistency when input to hydrus2d.
Hope this helps.

## ID = 441, How to deal with 3-dimensionality

Wibri, 09/06/2005
Hello all,
I am looking for references or thoughts about how to relate results from Hydrus 1D or 2D to 3D field results.

**ID = 442, Atmospheric BC with runoff**

Paul Thomas, 09/11/2005
I am simulating flux through a soil column and have had trouble getting Hydrus to generate runoff. I'm using a precipitation data set that includes rainfall events that should result in significant runoff volume. I understand that the atmospheric boundary with runoff should allow infiltration up to the capacity of the upper soil material, then dump the excess as runoff. I've increased discretization density in the upper profile and used initial and minimum time steps down to the one-second range. I've even artificially decreased the permeability of the upper soil...still I get 100% infiltration. Any other suggestions?

Jirka, 09/12/2005
What is the relation between the precipitation rate and the saturated hydraulic conductivity?
J.

Paul Thomas, 09/12/2005
The upper layer has a hydraulic conductivity of 4.8 cm/day. The precipitation rates routinely exceed 5 cm/day and peak around 10. vTop and rTop are still equivalent. The upper layer is only 2.4 cm thick and is underlain by a layer with a saturated hydraulic conductivity of about 55 cm/day.

Jirka, 09/12/2005 :
Then you do not get any run off since the infiltration capacity is larger than the precipitation rates. Infiltration rate can be for a short time larger than Ks, see typical infiltration rates. THe underlying high conductivity layer also increases infiltration capacity, since it increases pressure head gradient. Thus I would not be surprized that you do not get any run off.
Jirka

Paul Thomas, 09/12/2005
I can see now that the underestimation of runoff is due to spreading out precipitation over a 24 hour period. I can impose a correction on precipitation based on real runoff data. Thanks for your help.

**ID = 443, Import output from previous simulations**

Adsmith,09/13/2005
Hello,
I would like to know if there is a way to import pressure head output files from previous simulations as initial conditions for a subsequent simulation. I am trying to obtain a steady state solution for a 10 m column with an upper clay layer 2.4 m thick and a lower sand layer 7.6 m thick. I am finding that excessive run times are becoming a problem due to the large number of nodes and the very small time steps

required to obtain convergence. I am hoping that by importing pressure head profiles from intermediate times I may be able to solve this problem.

Thank you.


Mvcallaghan,09/14/2005
adsmith,
I'm not sure if it is possible to "hotstart" simulations in Hydrus 1D.
The low-tech way of importing the final pressure profile into Hydrus is as follows:
1) open the NOD_INF output file in Wordpad.
2) copy, paste and save (as a new text file) the timestep you are interested in importing
3) import the new text file into MS-Excel
4) copy and paste the presure column from Excel into the Soil Summary in Hydrus


adsmith,09/14/2005
Thank you for your reply, that was very helpful.


Jirka 09/22/2005
Adsmish,
The suggestion given above is correct. I usually open the Nod_inf.out file directly with Excel and then get the required profile for a selected time.
I have implemented import of intermediate pressure head (concentration or temperature) profiles as initial condition for a new simulation in HYDRUS-2D, since there it would be rather difficult to do that manually (for complex geometries). In HYDRUS-1D, I have found it so easy to do that manually, that I did not implement that into the software.

Jirka


**ID = 444, HP 1 release**

s.zanarello,09/20/2005
Hi everybody
are there same news about HP1 (hydrus-PhreeQC) release?
Thank you very much


Jirka,09/22/2005
HP1 was released in August 2005. You can get all information online at: http://www.sckcen.be/hp1
I urge everyone to check out that web site. HP1 is an increadibly powerfull tool (but complex). Here is a brief description:
HP1 - A Coupled Numerical Code for Variably Saturated Water Flow,
Solute Transport and Biogeochemistry in Soil Systems.

The HP1 code was recently developed to simulate one-dimensional variably saturated water flow and multicomponent transport in soil systems and sediments for a broad range of low-temperature biogeochemical reactions in water, soil and sediments, including aqueous speciation, cation exchange, mineral dissolution/precipitation, and (bio)degradation for mixed equilibrium-kinetic systems.

Any combination of the following features can be described by the HP1 code:
* One-dimensional transient water flow for different boundary conditions including atmospheric conditions (precipitation, evaporation, transpiration)
* Root water uptake as a sink for water
* Root growth
* One-dimensional transient convective and conductive heat transport for time-variable temperatures at the soil surface
* One-dimensional advective, dispersive and diffusive transport of multiple solutes
* Effect of temperature on transport parameters, thermodynamic constants, and rate parameters
* Different functional forms for the soil hydraulic properties, including hysteresis
* Physical non-equilibrium solute transport
* Physical and chemical spatial heterogeneity
* Equilibrium aqueous speciation reactions and kinetically controlled aqueous reactions such as radioactive decay
* Multi-site cation exchange related to type and amount of minerals present
* Equilibrium and kinetic dissolution/precipitation of primary and secondary minerals
* User-defined kinetic reactions
* Simultaneous presence of different reactions (sequential and parallel kinetic reactions, equilibrium and kinetic reactions, homogeneous and heterogeneous reactions, biogeochemical reactions)


s.zanarello, 09/26/2005
Thank you very much
Stefano


## ID = 445, Simulation of a vertical drainage pipe

Sofia,09/21/2005
I am simulating a part of a landfill cover. The downslope boundary of this part is assumed fo be a vertical drainage pipe (i.e. a vertical perphorated pipe that collect the leachate). What type of boundary condition do you advise to simulate the effect of the drainage pipe.
Would a seepage face be a good choice?

Jirka,09/22/2005
Sophia,
If there is no standing water in the vertical drainage pipe, then seepage face would be a good condition for this purpose.
However, you need to realize that vertical pipes will make the problem 3D, that it is not any more a 2D problem.
Jirka

**ID = 446, Evaporation flux**

Sofia,09/21/2005
The output file Cum_Q.out shows the atmospheric cumulative flux (CumQA), i.e. the algebraic difference between evaporation and precipitation . How can I get the actual volume of the EVAPORATED water? Do I have to calculate the amount of precipitation and substsract it from the atmospheric flux? Is there an easier way?
One more question, that may seem silly, how can I import a column data from an output file to an excel worksheet.
Thanks
Sofia


Jirka 09/22/2005
With 2D, there is no easier way at the moment. In the next version (coming in January 2006), I will separate in the postprocessing the evaporation and infiltration fluxes. By the way, HYDRUS1D does it already now.
This output file is ASCII and thus it is easy to read (open) it directly with Excel.
Jirka


**ID = 447, About initial conditions**

EfeT 09/21/2005
Hi,
Thanks for previous help rendered to me through this forum.
I am running a simulation of water flow,and i assigned constant pressure head value as 0.28(based on TDR measurement from physical model),the water flow parameters(Qr=0.1288,Qs=0.4569,alhpa=0.0453,n=2.4364,Ks=201.99 and I=0.5) were obtained through the neural network predition.The simulation yielded a result that initial condition is either smaller than Qr or larger than Qs.
As a new user i could not trace my mistakes,kindly help me out.Also i would like to know the maximum observation nodes one can insert in a domain.

Thanks.
Efet


Jirka 09/22/2005
Efet,
I can not see any reason why the code would not accept a water content of 0.28. The code only checks that this value is larger than Qr (residual water content) and smaller then Qs (Saturated water content). Obviously TDR measures water content, while you are saing that you assigned "constant pressure head". Misprint? Make sure that you are specifying the initial condition in water contents and that indeed the initial water contant is in the interval between Qr and Qs for all soil horizons.

Maximum number of observation nodes. This depends on the version you have. Initially I allowed only 5, but then since users complained about it, I kept increasing it. Keep on entering observation nodes, until the code tells you that you can not enter

any more. Code will however display the output only for 10. If you enter more, then you need to do your own graphs.

Jirka


**ID = 448, Inverse modelling using more than one solute**

Dan Dolmar, 09/22/2005
We have a need to optimize a reaction chain model. We want to have the model estimate degradation rate coefficients between contaminants in the reaction chain.

The new HYDRUS-1D v3 does not appear to have the capability to use concentration data from more than one solute in the inverse problem.

1) Is that correct?
2a) If the answer to 1) is "yes", is there some clever way to run two separate inverse problems so that the reaction parameters derived from solute A data will "agree" (to whatever extent) with the reaction parameters derived from solute B data (assuming that A degrades to B)? It seems to me that the deg rate from A to B that was estimated using only Solute A data could be very different from the deg rate from A to B that was estimated using only Solute B data.
2b) If the answer to 1) is "no", what am I missing? Where do you specify which solute you are entering data for?

Thanks folks!
--Dan


Jirka, 09/22/2005
Dan,
HYDRUS-1D actually does allow you to optimize parameters for two solutes, and that can be then used to better define degradation coefficients. But this option is not documented in the manual and not advertized.
See, for example, the following paper, for which I originally developed this capability:
Casey, F. X. M., and J. Šimùnek, Inverse analyses of the transport of chlorinated hydrocarbons subject to sequential transformation reactions, J. of Environ. Quality, 30(4), 1354-1360, 2001.
Thus:
1) No, HYDRUS can use two solutes in the inverse option.
2) In the Inverse data dialog, you will specify position for solute 1 as usual (i.e., position is equal to the observation node number), for solute 2 you will make this position negative (i.e., position is equal to the negative of the observation node number).

Jirka

Julie 09/04/2006
Hello folks,
I am running inverse procedure to find adsorption parameters which can reproduce my column measurements.
If I use one solute, everything is alright.

But if I want to add one more solute (solute transport - general information window) to simulate mineralization of the first solute (with a reaction chain model), my hydrus says : "inverse solution is implemented only for one solute"

What can I do?
Thanks,
Julie


Jirka 09/04/2006
Julie,
Do you have version 3.0 of HYDRUS-1D? I do believe that this version should allow to simultaneously optimize parameters for two solutes as described above. You can download it from this web site.
Jirka


Julie 09/08/2006
Hello,
Thank you: the new version of Hydrus1D has a lot of new options, it is very appreciable!
However, I still have an interrogation: how can I differentiate the concentration measurements of solute 1 from those of solute 2 in the "Data for Inverse Solution" window?
Or: if I choose to use just the concentration of solute 2 for the inverse procedure, how Hydrus could understand that?

Regards,
Julie


Jirka 09/08/2006
Julie,
In the Inverse data dialog, you need to specify position for solute 1 as usual (i.e., position is equal to the observation node number), for solute 2 you will make this position negative (i.e., position is equal to the negative of the observation node number).

Example for two solute with one observation nodes (BTC):

Time Concentration Type(conc) Position Weight
139 0.00147417 4 1 1 BTC of Solute 1
144 0.0013792 4 1 1
149 0.00114506 4 1 1

154 0.00116869 4 1 1
159 0.00102837 4 1 1
.
.
.
29 0.00196186 4 -1 1 BTC of Solute 2
34 0.00348265 4 -1 1
39 0.0123307 4 -1 1
44 0.0644014 4 -1 1
49 0.0970549 4 -1 1
54 0.132315 4 -1 1

Jirka


Julie 09/08/2006
Many thanks folks. I hope you do not feel so repetitive.
Julie


### ID = 449, Two kinetic sorption sites for colloid release

cobaltblue 09/22/2005
hello Jirka
Questions are:
1. In the Attachment-Detachment Model,how to difine the initial condtion at sorption sites S1 and S2 when both sites in solid phase are not equal to zero and what is the fraction between two sites.
2. I found that the detachment coefficient for the Second sorption site did not work in the direct method when attachment coefficient for the Second sorption site was set to zero(blocking was not considered ). I want to know this happen.


Jirka, 09/22/2005
1. The code allows you to specify initial concentrations for the S1 sorption sites. It does not allow you to do that for S2 sites, it assumes that initial concentrations at S2 sites is equal to zero.

One does not need to specify the fraction between S1 and S2 sites. Sorption to both sites is handled independently through their attachmment and detachment coefficients.

Sorption concentrations on S2 sites is printed into the last column of Nod_inf.out file (not displayed in the interface).

2. When ettachment coefficient for S2 sites is equal to zero, and since the code assumes that initial concentration at S2 sites is equal to zero, then obviously there is nothing to detach, irrespecitve of what is detachement coefficient.
Jirka

**ID = 450, Desorption of pesticides**

Pancho, 09/26/2005
Hello,
I am working with transport of pesticides, and I did miscible displacement experiments using soil columns. I fit a chemical nonequilibrium model to the observed values, and i get good results, but i dont know how i can consider the desorption.
i have values of Kd obtained from batch experiments and values of the dispersivity obtained by tracer experiments. I fit the value of F and alpha, i think that alpha its related with the desorption, but dont know how. please, anyone can help me?
now im doing some batch experiments to obtain the desorption coefficient independently from the displacement experiments, so, how can i consider that fact and not adjust this parameter?

Thanx
Francisco Suárez


Jirka 09/26/2005
Francisco,
You are using a two-site sorption model, which assume that fraction of sorption sites have equilibrium sorption, while another fraction of sorption sites have kinetic sorption.
When you compare the kinetic sorption equation used in HYDRUS (first order kinetics with kinetic constant omega) with attachment/detachment model (i.e., sorption rate/desorption rate) you will see that desorption rate, k_d is equal to the first order rate coefficient omega, and that the sorption rate, k_a is equal to disorption rate k_d * bulk density / water content * distribution coefficient K_d
k_a = k_d * ro / theta * K_D
k_d = omega
This is, however, derived for one-site kinetic sorption.
Jirka


Pancho, 09/27/2005
Hi Jirka,
I look at the attachment/detachment model and i dont understand what does it mean the dimensionless colloid retention function...
I compare the attachment/detachment model to the kinetic sorption used in HYDRUS and i concluded that K_a = K_d * ro/theta * (1-f)*K_D and K_d = omega for the two site model (considering the dimensionless colloid retention function equal to 1). Is this right? ... the value of omega is the same value of alpha?
Please, can you tell me some references that i can read to understand this better?
Also, i have to put variable flux on hydrus 2D (on the top of the domain), but i dont find where i can put the data for the variable flux. I put variable flux using the atmospheric condition, but i want to know how if can i use the variable flux without using the atmospheric boundary condition.. it is possible? or the only way to put the variable flux is the atmospheric condition? (in HYDRUS 1D i use a window that allows me to put the variable flux, but i dont find this on Hydrus 2D)

Thank you very much for your help! =)


Jirka, 09/27/2005
Francisco,
Your derivation is the same, except that you did consider the fraction of kinetic sorpion sites.
Dimensionless colloid retention function: This takes care of possible filling of the sorption sites, i.e., when a lof of colloids is sorbed, there is no place for more colloids to get attached. For pesticides, this may not be relevant.
I have not seen comparison of these two models in the literature. But they are mathematically identical, as you could see during your derivation.
Variable flux: In HYDRUS-2D you can use either atmosperic BC (Prec column) or time-variable flux BC (rGWL column). You need to click on "Time Variable BC" in "Time Information" dialog window.

Good luck
Jirka


Pancho, 10/02/2005
Thank you a lot Jirka, now i think that i am in the good way!


Srilert, 01/23/2006
Dear all,
I read the former post about desorption and i try to read manual but i cannot understand them. Could u pls. tell me more details about desorption that u can simulate heavy metals like Pb from the top of surface into ground water in case 1) no rainfall and 2) rainfall
In my case, Can use like this above case (Pesticide desorption) ?
Could anyone help me pls.?
Thanks all in advances.
Srilert


**ID = 451, How to set cRoot for turgrass with ammonium and ni**

Rainfields, 09/26/2005
How to set cRoot for turgrass with ammonium and nitrate?
Can anybody help me?

cRoot Value of the concentration for the fifth time-independent boundary condition [ML-3]. If water uptake is considered then cRoot is automatically used for the maximum concentration of water removed from the flow region by root water uptake. When zero is specified, then all solute is left behind in the soil and only solute-free solution is being taken up. When the concentration is lower than cRoot, all solute is taken up. When the concentration is higher than cRoot, the excess solute stays behind.

Set equal to zero if no fifth time-independent boundary condition and no solute uptake by roots is considered.

Thanks,
rainfields


Jirka 09/26/2005
rainfields
This will depend on the nutrient demand by crop. If you think that plants can take all nutrients they can get, them make cRoot large for both ammonium and nitrate. Then plants will take whatever is dissolved in water they use for transpiration. You need to find out in the literature what is the nutrient demand. If it is smaller than how much plants would take, then you need to decrease cRoot.
Jirka


**ID = 452, Flux at interface**

sofia 09/26/2005
Dear Jirka
I am modeling a two-layers system. Is there a way to find the percolation across the interface?
Thanks
Sofia


Jirka, 09/26/2005
Sophia,
Regular HYDRUS-2D code does not calculate fluxes across cross-sections within the transport domain. However, I do have a version which can do that and will email it to you. This feature is, however, not supported by the graphical interface and needs to be handled manually.
Jirka


**ID = 453, Flow equation solution**

Adsmith, 09/28/2005
Hello,
My question is about how Hydrus solves the modified form of the Richard's equation. This equation incorporates terms for both pressure head and volumetric water content, which are both unknown. In my initial conditions, I specify a pressure head profile. Does this mean that Hydrus takes my pressure head values, incorporates them into the Richard's equation, and then produces the corresponding volumetric water contents? Basically, I would like to know which term Hydrus treats as the primary variable: pressure head or water content?

Thank you,
Amanda

Jirka, 09/28/2005
Amanda,
The detailed description of the numerical procedure is in the manual.
Pressure head is the primary variable.
Jirka


## ID = 454, Runoff simulation

s.zanarello, 10/06/2005
Hi
I want to evaluate runoff using Hydrus-1D. I know that Hydrus treats as runoff all the water that exceed the infiltration capacity. I also know the in real world runoff is a function of surface acclivity.
My question is: in real world, is the infiltration capacity a function of surface acclivity? When I have runoff (in a slope), is the superficial layer of soil always satured?
If infiltration capacity is not a function of surface acclivity, why runoff amount is a function of this factor?
Thanks and by


## ID = 455, Injection well – boundary.out

Franta, 10/18/2005
I have two questions related to the injection well and BOUNDARY.OUT file

A) I have used an axisymmetrical type of flow for an injection well simulation. I have set up a RECTANGULAR mesh. I thought that it is going to be OK if I neglect the well itself since the radius is so small (R = 7.6 cm). I have divided measured injection rate between individual nodes that represent a screened segment of the well (10 nodes on the vertical wall - top to bottom 4.6 m) by specifying a NODAL RECHARGE for those 10 nodes. Hydrus correctly assigned a CONSTANT FLUX boundary to the segment when checked it in the Boundary Conditions editor. When I have selected constant flux and double clicked on the individual nodes I have received ZERO FLUXES for all nodes. Is it because there is no area that Hydrus can divide the NODAL RECHARGE?
I have also checked the BOUNDARY.OUT file and found out that NODAL FLUX values are missing for all print times. Is it because those selected nodes represent a flux for a surface area of a cylinder? I have also expected that Constant Flux will be flat; however, it decreases exponentially over time. Cumulative Constant Flux has not been linear either.
Other than that simulation has converged.

B) Since A) have not worked the way that I have expected I have set up a GENERAL mesh with an axis going thru the middle of the injection well (R = 7.6 cm). I have specified an individual NODAL RECHARGE as specified in A). When I selected constant flux and double clicked on the individual nodes I have received fluxes for the top most and bottom most nodes that were double of that for any node in-between (66.1785 and 33.0893). Should not the outer nodes have a half of the flux?

I have also checked the BOUNDARY.OUT file and found out that NODAL FLUX values are missing for this geometry as well.

Constant Flux was not again flat and decreased over time. Cumulative Constant Flux was linear over time as I have had expected.

This run has converged as well.

Is B better than A?

Thanks a lot

Franta

Jirka, 10/22/2005

Franta,

Could you zip both folders with their respective *.h2d files and email it to me? From your description it is not clear to me where the problem is. I will check out what you have done and where the problem is (if there is one).

Jirka

## ID = 456, MIM

Jtourneb, 10/19/2005

Dear All

In the context to reproduce experiment in 1D column (tracer Chloride and Isotope), near saturation, in a sandy loam soil, we want to use MIM option. Because elution curve is not symetric showing some exchanges, we also showed anionic exchange, simulated by a negative Kd.

Hence, in case of physical non equilibrium, Hydrus 1D propose the relationship 3.18 teta=tetam+tetaim, can we have access to the repartition, because in Solute reaction Parameters, option Thlmob is not available and has no effect.

We accurately ajusted for Chloride (with frac, thlmob, kd and alpha), but it was not good for isotope, despite a similar BTC (in shape).

We use inverse method to fit parameters : frac, thlmob, kd and alpha, without satisfactory results.

If you need, we can send project.

Julien

Jirka, 10/21/2005

I'm not sure that I follow your question. What do you mean by "in Solute reaction Parameters, option Thlmob is not available and has no effect".

In the direct mode, ThIMob is in the "Solute transport parametes" dialog window, while in the inverse mode all solute parameters are available in "Solute reaction parameters" dialog window. If ThImob>0 then it surely must have an effect. It may not be sensitive to your data, but that is another question.

Jirka

Jtourneb, 10/22/2005
Hi
Indeed I understand that normally we can play with this option, but actually it is not possible.
If we use inverse methode, thmob remains the same, without any convergence, and for direct method, thmob is replaced by "Not Used".

Otherwise, can you confirm that frac and thmob are different: frac concerns for sorption; and thmob for mobile immobile partition?
Regards
Julien


Jirka, 10/22/2005
Julien,
You can simulate either physical (MIM) or chemical (kinetic sorption) nonequilibrium with HYDRUS-1D. Theta_imob is not used when chemical nonequilibrium is simulated. I would bet that you are using colloid transport model that involves chemical nonequilibrium (attachment-dettachment), right? Then you can not have physical nonequilibrium at the same time.

Jirka


## ID = 457, Modelling phosphorous using HYDRUS 1D

Jaboun, 10/19/2005
I am trying to model PO4-3 transport using HYDRUS-1D in 1D soil columns. Does that fall under the category "Solute transport>general solute transport"? What type of adsorption isotherm will that option utilize?
thank you.


Jirka, 10/21/2005
When you use "General Solute Transport", HYdrus will allow you to use:
a) no adsorption
b) linear adsorption isotherm
c) Freundlich isotherm
d) Langmuir isotherm
e) Freundlich-Langmuir isotherm

Jirka


## ID = 458, Subsurface drip irrigation design

thabo23, 10/24/2005
Jirka,

I have used Hydrus 2D for furrow irrigation with success.

Now we have a pasture system and trying to see the suitability of a subsurface drip irrigation for a variety of soils under pature. I am wondering wheather Hydrus 2D can be used to get some ideas for designing of the sub-surface drip irrigation system? If yes, where do we need care?

Your suggestions would be much appreciated.

Thanks

Thabo

Jirka, 10/24/2005

Thabo,

There has been a lot of studies that used HDYRUS-2D for subsurface drip irrigation. Let me give you just some selected references that I can quickly find references to.

Best regards,

Jirka

Lazarovitch, N., and J. Simunek, U. Shani, System dependent boundary condition for water flow from subsurface source, Soil Sci. Soc. Am. J., 69(1), 46-50, 2005.

Gärdenäs, A., Hopmans, J. W., B. R. Hanson, and J. Simunek, Two-dimensional modeling of nitrate leaching for various fertigation scenarios under micro-irrigation, Agric. Water Management, 74, 219-242, 2005.

Meshkat M, Warner RC, Workman SR, Modeling of evaporation reduction in drip irrigation system, Journal Of Irrigation And Drainage Engineering-ASCE, 125 (6): 315-323, 1999.

Assouline, S., The effects of microdrip and conventional drip irrigation on water distribution and uptake, Soil Sci. Soc. Am. J., 66, 1630-1636, 2002.

Schmitz, G. H., N. Schutze, and U. Petersohn, New strategy for optimizing water application under trickle irrigation, J. of Irrigation And Drainage Engineering, 128(5), 287-297, 2002.

Cote, C. M., K. L. Bristow, P. B. Charlesworth, F. J. Cook, and P. J. Thorburn, Analysis of soil wetting and solute transport in subsurface trickle irrigation, Irrig. Sci., 22, 143–156, DOI 10.1007/s00271-003-0080-8, 2003.

## ID = 459, Water repellent systems

Cwillson, 10/24/2005

We are trying to use Hydrus 1D to model infiltration and drainage into water repellant systems (for the time being we are disregarding preferential flow paths). However, are having some problems getting the model to converge when modeling water infiltration using "higher" VG n values (e.g., greater than 7). As expected the "drainage" VG parameters that we measured in the lab are not impacted by the water repellancy, but the "imbibition" VG curves are.

Any comments/advice on what we can try OR literature that may help us would be appreciated.

Jirka 10/24/2005
Large n values are characteristic for monoporous systems, i.e., sands and gravel. Then water content decreases dramatically within very narrow range of pressure heads (at the retention curve). This results into very steep infiltration fronts. Consequently, you need to have a very fine spatial discretization to accomodate these steep water content fronts.
Jirka

Cwillson, 10/24/2005
Thanks for the quick response! In addition to the water repellancy, we are also using a uniform medium/coarse sand (compounding the problem---not always the easiest way to start ). We will refine some more and see what happens.

## ID = 460, Infiltration in 1D columns

Jaboun, 10/26/2005
Hello,
I am simulating solute transport (phosphorus) using HYDRUS1D, but i am having problems getting the model to converge.
I have clay and silty loam clay soils; i have three layers of soil material. And the columns were saturated frm thebottom up; so they head on top.
I think the problem is with the water flow
1) I chose an upper BC to be Atm BC with surface layer because i am considering infiltrating and evaporation, hence i needed to chose a variable time boundary. With that option the model did not converge

2) then i chose the upper BC to be constant and specified the pressure head to be equal to 8 cm on top of the soil at the observation nodes, since i had a head above my columns. The model did converge but it gave me a water mass balance error between 100 and 140 %. That is too mcuh.

What do you suggest i do?
Thank you.
Joumana

Jirka, 10/26/2005
Look at the example Test1 in the direct worskspace on what discretization should be used.
When you have clay in the profile, use soil hydraulic model with -2 cm air entry.
Jirka

Jaboun,  10/27/2005
Thank you for the quick response, Jirka. Combining examples 'test1' and 'test4', i got the model to converge.
Thanks again.
joumana


jaboun, 11/08/2005
I would like to ask about the term "pusle duration" (solute transport module). Based on the definition spcified in hydrus, it is basically the time of the concentration pulse, duration concentration change across boundary or between nodes or across the whole region of study
Does the value we specify relative to units of time we sepcify as well (minutes, hours, or days)? And why when it is reported a time is specified next to it hours (test4 in manual pg. 134).

It is still not clear to me how it can affect the results.
Thank you.
Joumana


Jirka, 11/09/2005
Pulse duration is in time units, in the same time units as all other time-related variables,
Jirka


## ID = 461, High n alpha values

Zaydoun, 10/26/2005
Hello;
I am using Hydrus-1D in simulating 100cm long column filled with coarse/medium particle size and with the following soil hydraulic properties: n=10.57, Theta_irr=0.016, Theta_sat=0.348, alpha=0.0995 1/cm and K_sat=29.55cm/hr. The model didn't converge, although I made many trials in spatial and temporal discretization changed as well as iteration criteria.
Thank you
Regards
Zaydoun


## ID = 462, Unsaturated hydraulic conductivity vs time

Preecha, 10/27/2005
Dear HYDRUS Users,
I wonder how I can obtain unsaturated hydraulic conductivity of each subregion for each time interval (or time step) from the simulations results.
Kind regards,
Preecha

Jirka, 10/27/2005
At each print time, the code prints the average pressure headfor each subregion into the balance.out output file. If each subregion is homogeneous, then you can convert it into hydraulic conductivity using soil hydraulic parameters.
Jirka

## ID = 463, Free water surface

mizo316, 10/28/2005
What is the best way to model free water in a ditch? a constant pressure head on the surface?
Thanks

Jirka, 10/28/2005
Use the option "Constant head with equilibrium from lowest located nodal point"
J.

## ID = 464, Parameter "I" in soil hydraulic properties

Edsrlima, 10/28/2005
Hello,
I'm simulating unsatured water flow in a column and I want to know the meaning of the parameter "I" in soil hydraulic parameters ?
If is possible could you give me some default values for this parameter?
Thanks,
Edson.

Jirka, 10/28/2005
Edson,
That is the tortuosity parameter in the hydraulic conductivity function. Unless you have unsaturated hydraulic conductivity data (which most people don't), you can specify this parameter equal to 0.5 as a first approximation (Mualem, 1976).

Jirka

## ID = 465, Using deep drainage BC

Bndiaye, 10/31/2005
Hello everybody
I am modelling a rice pad with a 10 cm pressure at the surface (irrigation by flood waters) and deep drainage to a surface aquifer at 150 cm from the surface (position of the aquifer in the deepest state as measured in piezometers)

My questions are:

1- How to estimate the parameters A and B in the formula given in the manual? Is it realistic to use the same parameters as in examples problems?

2- As the aquifer is replenished by the surface water by infiltration, how does the program calculate deep drainage flux if there is a hydraulic continuity between surface water and the aquifer?

Jirka, 10/31/2005
1. See Hopmans and stricker (1989) and references in this paper. Hopmans, J. W., and J. N. M. Stricker. 1989. Stochastic analysis of soil water regime in a watershed, J. Hydrol., 105, 57-84.

2. If the entire profile is saturated, the code will assume that the water level is at the surface or above, depending on the pressure head profile. The code considers the pressure head at the bottom of the profile in the deep drainage recharge equation.

Jirka

**ID = 466, Simulating water flow in a column**

Edsrlima, 11/03/2005
Hi,
I'm simulating water flow in a soil colum. These are the soil hydraulic parameters: Qr=0 cm3/cm3, Qs=0,443 cm3/cm3, alpha=0.0449, n=3.6732, Ks=16,18 cm/h. The colum have a depth of 70cm and a diameter of 5 cm. The initial water content is 0.0216 cm3/cm3. I hava a drip with rate of 150 cm3/h and the period of simulation if 3 hours. I want to use 10 nodes.
I'm having problems on how can I input de boundary and initial conditions. Could you help me?
Thanks,

**ID = 467, van Genuchten**

Sparks, 11/10/2005
I have started using Hydrus 2-D and am needing some help understanding van Genuchten's equation. The help button in the soil hydraulic model section, says that Se is the effective water content. However, when I look through van Genuchten's paper he calls S the absolute value of the slope of capital theta with respect to log h. And then, when I searched the internet, another site called Se the degree of saturation.

My main concern is: what IS Se? And how is it calculated?

Rvang, 11/11/2005
Sparks:
Se is effective saturation, as given by Eq. (2.17) in the HYDRUS-2D manual, or Eq. (6) in the RETC manual. We used Cap Greek Theta for Se in the original 1980 paper, while S was indeed used for the slope of Theta (or Se) versus (log h). S or Sw is often used for relative saturation (or sometime degree of saturation), without the residual water content in there. Still, you do not have to be worried too much about all of this since Se is not an input parameters but internally used to calculate the hydraulic properties from the input parameters as specified under "Water Flow - Soil Hydraulic Parameters".

Hope this helps. --Rien van G.


**ID = 468, SWC**

mizo316, 11/13/2005
If the hysteresis option is not selected. Would the drying cycle in hysteresis is selected as default?
Thanks.


rvang 11/13/2005
Mizo,
If you do not select the hysteresis option, the program uses whatever hydraulic properties you specify in "Water Flow - Soil Hydraulic Parameters", which can be drying, wetting, or whatever you want. Only when you select hysteresis will alpha be for drying and you need an additional parameter (alpha_w) for the wetting curve.

It was great seeing you in Salt Lake! --Rien.


**ID = 469, Orthomin terminates, too many iterations**

Heather, 11/14/2005
Hello,
I've been trying to reproduce a simulation of drainage on a hillslope and am having difficulty getting it to run, without getting the "ORTHOMIN – TOO MANY ITERATIONS" comment.

The slope is15 m by 1 m and slope angle is 40°. The lower boundary has a no flow boundary condition, the toe of the slope has a seepage face boundary, and the upper surface has an atmospheric boundary. However, there is zero precipitation and evaporation in the atmospheric file (maybe there is a more appropriate upper boundary condition that I could be using?). The entire region is defined as a sandy clay loam, using the van Genuchten parameters from the Hydrus database. The system is also initially saturated, so the initial water content is 0.39. I'm assuming that this could be the reason for the instability. I am running this with the van Genucthen – Mualem hydraulic model with air-entry value of –2 cm checked.

I've tried increasing mesh spacing several times, starting with 1 cm and increasing to 10 cm spaces, but this has not fixed the problem. I've also tried reducing the minimum time step to 0.0001 days and increasing the number of iterations to 50.

Do you have additional suggestions on ways to improve the problem? I would really appreciate any advice.

Sincerely,
Heather


Jirka, 11/14/2005
Heather,
Use pressure head as initial condition and leave part of the profile at least partially unsaturated.

J.


Linlin, 01/31/2007
hello!
i met the similar problem, and i used pressure head as initial input and unsaturated as the initial condition. under heavy storm about 14cm/d, i want to simulate interflow. but the calculating process can't go on. i think the region should be partly saturated.
do you have good suggestion?
thanks!


**ID = 470, Simulating runoff**

Koenverbist, 11/14/2005
Dear Jirka,
I am trying to evaluate runoff harvesting techniques with Hydrus2D. The problem is that runoff is immediately removed from the simulation. However, the infiltration trench I am trying to model receives this runoff, and could be seen as an increasing variable pressure head boundary. After the rain event, the accumulated water infiltrates, turning it into a decreasing variable pressure head boundary.

Is there any way to simulate filling up of the trench using excess rainfall intensities, or will I have to calculate the runoff first and apply this as a variable pressure head boundary on the sides of the trench in a subsequent simulation.

Any help on this issue or any similiar studies performed by other researchers using Hydrus2D might help.

Kind regards,
Koen.

Jirka, 11/14/2005
Koen,
With the standard HYDRUS-2D version that is the only option you have, i.e., "to calculate the runoff first and apply this as a variable pressure head boundary on the sides of the trench in a subsequent simulation."

I can see that one could develop a special BC where one would integrate access water (different between precipitation and infiltration) and divert it into furrow. One would then keep a mass balance of this water (runoff, infiltration, storage change) and calculate from it the water level. But that seems too specialized to spend much time on it and thus your simple approach seems more realistic.

Jirka

## ID = 471, Nodal recharge

Edsrlima, 11/14/2005
Hi,
I want to simulate water flow in a soil colum. The soil hydraulic parameters obtained with van GENUCHTEN (1980) model are: Qr=0, Qs=0,443 cm³/cm³; alpha= 0,0449; n=3,6732; m=0,1119. My colum have 5 cm of diameter and a depth of 70 cm. I am using 10 observation nodes along the columm and I have a emitter with flow rate of 150 cm³/h. I have a initial water content of 0.0216 cm³/cm³ and I put this boundary conditions: Lower BC is a Free Drainage and the Upper BC is a constant flux= 7.64cm/h (150cm³/h/19,64cm²). I don´t know why but the nodal recharge is changed (WHy?? Whats the meaning of nodal recharge?) when I plot my BC's and I can only simulate with a time that is larger than 8 hours.
When I plot the graphical results of my simulation I observed that the saturation level is not reached (Q=0.35 cm³/cm³).
Can you help me??
Thanks,

## ID = 472, Error message, unit 70

Franta, 11/16/2005
I am trying to optimize Alpha, N, and Ks using inverse solution and keep getting following message after first or second iteration: "File name missing or blank – please enter file name UNIT 70?" What is wrong? Thanks

Jirka, 11/18/2005 : 17:26:07
It is probably caused by the ORTHOFEM solver (I have actually discovered that myself this week) that does not converges and tries to send message about it to the output file. However, the output file into which this information is usually sent (during direct runs) is not open during inverse run (only for the final run). Just type any name and the code should continue. I will send you an updated code that takes care of this problem.
J.

Franta, 11/20/2005
Thanks, it worked!

**ID = 473, Problems with time variable boundary conditions**

Mary, 11/17/2005
I am trying to simulate flow in a one dimensional column with falling head. I assigned a variable pressure head at the top and specified two head values for time-varible boundary condition and my model convereged, but i realized that does not take into account the flux change when the head disappears and the pressure on top becomes less than zero. So i changed my top boundary conditions into variable pressure head/flux, but my program did not converge.

May you please help on that matter.
Thank You.

Jirka, 11/18/2005
YOu need to specify initial condition in the top node corresponding with the water level and select atmospheric boundary condition with surface layer. The code should lower the water level as water infiltrates into the soil profile.
Jirka

Anafiz, 01/26/2006
Thank you Jirka
That just solved it.

Anafiz, 01/26/2006
I am sorry. My reply "Thank you Jirka
That just solved it." was mistakenly posted here

Mary, 02/06/2006
Hello Jirka,
I have selected atmospheric boundary condition with surface layer, as you suggested, to simulate flow in one dimensional columns with falling head. And i have selected hCritA to be equal to 15000 cm. The problem i am having is that after the head of water on top of the column falls to zero, my mass blance error increases from zero to 50%. That is as the soil is drying, going from saturated to unsaturated, i am having a high mass balance error. What do i need to do to fix it, or what am i doing wrong. Pleas help me on that matter.

Thanks a lot for your help.

Jirka 02/06/2006
I have just emailed you an example that has 0.000% mass balance error. Compare this example to your problem and try to figure out what you do wrong.
Jirka

Mary, 02/13/2006
Hello Jirka,
Based on the model you sent, i was not able to make my solution to converge and decrease my mass balance to less than 1%. Just because i was my columns were saturated from the bottom up with a head of water on top; HYDRUS had a problem going from saturated to unsaturated conditions, it was very sensitive to hCritA. But I was able to successfully solve my problem with the dual porosity (Durner-dual van Genuchten-Mualem) hydraulic model. Why is that? Is that right?
Thank you.

## ID = 474, Residuals – FIT.out

Franta, 11/23/2005
I have used inverse solution to fit Longitudinal Dispersivity. I wanted to plot residuals against time, but "Residuals" column of FIT.OUT file had zeroes (.000) all the way from the top to the bottom. Have I done anything wrong? Should I look in different file to obtain residuals? Thanks

Jirka, 11/23/2005
Observed and fitted values that are used in the objective function, as well as residuals are indeed printed into the Fit.out file. You have done something wrong, or obtained perfect fit (unlikely), if residuals are all equal to zero.
Jirka

## ID = 475, Hydrus – PHREEQC

Srilert, 11/28/2005
I interesed in this model(hydrus-phreeqc) using in dissertation, is it available for download? could u pls. advice me ?

Jirka 11/28/2005
Lert,
You need to request it at the following web site:
http://www.sckcen.be/hp1/
Note that we are organizing an HP1 Course & Workshop Feb. 2006
http://www.sckcen.be/sckcen_en/activities/conf/conferences/hp1/index.shtml

Jirka

HYDRUS-PHREEQC-HP1 shortcourse/workshop to be held in Mol, Belgium, February 20-23, 2006. A unique feature of this meeting is that the new biogeochemical transport code HP1 (coupling between HYDRUS and PHREEQC) will be presented (day 3 of the training course). The HP1 code incorporates modules simulating (1) transient water flow in variably-saturated media, (2) transport of multiple components, and (3) mixed equilibrium/kinetic geochemical reactions. The course includes theoretical and hands-on computer sessions on vadose zone flow and transport (based on HYDRUS-1D), geochemical modelling (based on PHREEQC), and coupled reactive transport in variably saturated media (based on HP1).

The one-day workshop on "Modelling vadose flow and transport processes in radioactive waste management" aims to bring together current and future users of HYDRUS, thus providing an opportunity to show specific HYDRUS applications relevant to radioactive waste disposal and contaminated sites, and to discuss future needs and directions. The workshop will not exclusively deal with HYDRUS applications; other issues relevant to vadose zone flow and transport will also be addressed. We can still accept a few oral presentations, whereas posters are also welcome. The deadline for submitting your contribution is December 16, 2005. If you have any questions, please contact Monique Van Geel at mvgeel@sckcen.be or the HP1 website.

### ID = 476, Solute transport with degradation

Pancho, 11/28/2005
Hello,
I'm using Hydrus 1D to model herbicide transport on the vadose zone. I'm simulating very simple field conditions: constant precipitation of 0,3 mm/day; 0,5 mm/day and 1,0 mm/day. I put the groundwater level at a depth of 1 meter, and I run the program for 40 years. In the first 20 years I applied 2Kg/ha in one week of the year, then from the year 20 to the year 40 I run the model without solute pulse (wash).
I used a degradation coefficient calculated from the half life of the herbicide (mu = ln(2)/TD50). The adsorption coefficient its near to 1 cm3/gr and i used a two-site chemical non-equilibrium (alpha = 0.2 1/hr and F = 0.3 approximately).

when i look to the results i saw that the cumulative surface solute flux its correct, and in the 40 years i have 40 kg of herbicide mass. But when i look to the cumulative first order reaction [sum(cvCh1)], the mass that is removed from the system its more than 40 kg (about 50 kg for 0,3 mm/day; 45 kg for 0,5 mm/day; and less of 40 for 1,0 mm/day.. the last one its seem to be fine).

The sum(cvCh1) its the mass degraded from the system using the half life???
I saw the mass balance, and in there the concentration error its about 13%, and the water mass balance its very little (0.001%). Is something that i doing wrong?

Please help me!!
Thanx

Jirka, 11/28/2005
Francisco,
You need to use such spatial and temporal (time steps) discretizations, so that you keep the solute mass balance error below 1%. Thus you probably need to use finer spatial and/or temporal discretization.
Jirka


Pancho, 01/09/2006
thanks jirka, i change the spatial discretization and now its working very well


## ID = 477, 2D modelling

Srilert, 11/28/2005
I 'm not sure that I understand correctly. my question is
1. when I simulate in 2 dimentions problem, it mean type of flow both horizontal plane an dvertical plane?
2.in "water flow parameters" what does "I" mean causing I cannot find in help or manual?
3.in"Hydraulic model" how should i use , Van-Genuchen-Mualem or any others? any criterias ?

Thanks
Srilert


Jirka 11/28/2005
a) With a two-dimensional program you can simulate flow and transport in either x-z (vertical) or x-y (horizontal) plane
b) "l" (lower case L) is the tortuosity parameter in the hydraulic conductivity function.
c) Most people dealing with soil use van-Genuchten-Mualem model. Thus use it also.

Jirka


## ID = 478, Many elements of heavy metals in soil

Srilert, 11/28/2005
Dear all Hydrus 2d users
In my dissertation, I have 3 elements that concerned ,Pb, Mn and Fe in contaminated top soil and when acid rain flow throught the soil and then leach these heavy metal to unsaturated zone and throught groundwater. From above problem, I can use hydrus 2d simulate this problem or not.
And if it can do that, I have another question about "Solute Transport Parameters"
what can i find data base about "molucular diffusion coefficient in free water and in air"?
what does "Fract." mean?

If cannot, what models that can simulate properly?
Sincerely,
Srilert from Thailand


**ID = 479, Meshgen**

Srilert, 11/28/2005
Dear all that use Hydrus,
Can I use finite element programs that I have, to generate boundary condition and input into HYDRUS 2D? and how to do this?
sincerely,
Srilert


Mirek, 11/28/2005
Lert:
If you have Hydrus2D with Meshgen2D then you can do it. However, it is not a trivial task. You will have to create/modify file Domain.dat (in the Hydrus2D project subdirectory, see for example http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=409 for more information) and it means that you will have to get familiar with its format.

If you have Hydrus2D without Meshgen (Meshgen then runs as a demo version) then you will not be able to save projects with general geometry because only rectangular domains are allowed in this case.

Mirek


Srilert, 11/29/2005
I mean, I have another finite element program not meshgen. Can I use it crete boundary and input it in HYDRUS or not ?
Pls. let me know.
Regards,
Srilert


Mirek 11/29/2005
You can not use another program for Hydrus2D pre-processing (generation of the FE-mesh + input of boundary and initial conditions). That would be too complicated and time-consuming.
Mirek

**ID = 480, Transport in time-variable in unsat/saturated zone**

Jkrivic, 11/30/2005
Hi!
I'm trying to simulate solute transport with time-variable input function through partly saturated material with time-variable water table level. I would like to assign the time-variable solute input to just some of the cells on the top boundary.
I've used the variable flux boundary (with no solute flux) at the sides and the atmospheric boundary (with time-variable solute input) in the middle of the upper bounday. I've used the variable pressure boundary (with positive values) at the bottom of the cross-section in order to simlate oscillations of the water table.
That way I've managed to simulate the water flow succesfully, but I get the influx of the solute from the top and bottom boundaries simultaneously!
How can I assign the time-variable solute input for just a selected number of cells at the upper boundary in such case?
Regards,
Jure

Jirka 11/30/2005
Jure,
In solute transport "Time variable boundary conditions" you have an option to use up to 3 time-variable values - cValue1, cValue2, and cValue3. Thus you can use these three values and describe it at different parts of the boundary. I.e., cValue1 at the bottom, cValue2 at the top middle, and cValue3 at the top sides. Or you can use any combinations of these. You just need to specify pointers in the boundary module to point at these particular values.
Jirka

Jkrivic, 12/01/2005
Jirka,
Thank you for this explanation!
I've managed to use this "pointer" option without really knowing what it meant. There is no explanation in the documentation. Sadly, in contrast to the flow modeling there is not much advice to be found on solute transport simulations anywhere. So I have some more questions:
1. In the reaction parameters for solute menue there is a parameter Kd, but the help button displays a Ks parameter. Which parameter is this? And how are the other parameters used accordingly?
2. The solute transport parameters menue requires an input of Bulk density. What are the units of Bd when the unit for mass in the solute transport menue is mmol (which is not a mass unit anyway, but used as a default!)? Are the mass units defined some other way?
3. I have a stable model with very small water balance error but I don't really know how to achieve a sufficiently low concentration balance error in general?

Best regards,
Jure

Jirka, 12/01/2005

Jure,

1) Kd (and Ks in help) in solute transport parameters is the distribution coefficient that relates liquid and solid concentrations.

2) Bulk density [M/L3] must have exactly the same units as Kd [L3/M], since in the governing equation there is always a product of these two variables (leading to a dimensionless number). M in the units is not related to concentrations (e.g., mol, mg), but to M of soil [e.g., g or kg]

3) The only way you can improve solute transport mass balance errors is using finer spatial and temporal discretization. Using larger dispersivity may also sometime help. But, in general, it is mostly dependent on discretization of the spatial and temporal domain.

There is still a manual, which should be used as the main reference, and where everything is quite well explain (at least I think).

Jirka

Jkrivic, 12/05/2005

Jirka,

Thank you for your answer!

If the mass M in the units is not related to concentrations, but to M of soil, how, and in which menu is then the unit for concentration defined?

Jure

Jirka, 12/05/2005

There are tw types of M units, those that are related to concentrations (those are entered in the General Solute Transport Information Dialog) and those that are related to soil. It seems to me that by analysing the governing solute transport equation, it should not be difficult to figure out which are which. Bulk density and distribution coefficients (but also those that reactions constant that multiply bulk density) are related to soil, those that appear in concentration (but also partly distribution coefficient for Freundlich type adsorption isotherm) are related to solute.

Jirka

Jkrivic, 12/06/2005

Jirka,

Sorry for bothering you again, but I would really like to clear this before I do something silly with my model.

I would like to input the soil related mass in kg and the concentration related mass in mmol in the same model. Unfortunately, I did not find a way to define M units in concentrations and the soil related M unit separately. Can it be done at all? If not, than the unit for both mass quantities is the same, and is user-defined in the General Solute Transport Information Dialog. Am I missing something?

Jure

Jirka, 12/08/2005

Mass units in Solute transport dialog are meant to be for concentrations. It is simply a string (sequence of characters), which is inconsequential, except that it is printed to some graphs. Concentration can be given in any mass units (including dimensionless, g, mg, mol, mmol, etc). Note that concentration is in every single term of the governing equation (and thus it does not matter much what you use). Bulk density units (mass/volume) can be defined in different units (e.g., grams of soil, or kg). You have to only check that the distribution coefficient (that always multiplies the bulk density) is in the same units (the product of ro and Kd is dimensionless).
Jirka

Jkrivic, 12/09/2005

Jirka, thank you for your explanation!
Best regards,
Jure

### ID = 481, Sealing layer – max percolation

Uwe, 12/08/2005

Dear modellers,
I did some extensive modelling with HYDRUS2D of soil covers for waste sites. One results was, that a cover with a sealing layer could result in a percolation of > 100 mm/year. These findings are much higher than I expect, and I did some plausibility checks and have some questions...
I create a Hydrus2d-cover: two soil layers, the deeper has an ks 1x10-9 m/s.
Infiltration water on that soil cover, there develops a pressure head on that sealing layer up to the top boundary (left and right boundary: no flux).
Water is percolating through the sealing layer - everything looking ok.
BUT: The maximum pressure head on that sealing layer (thickness 0,4m) is 1,5m (the thickness of that soil layer). Calculating with DARCY:
max q = ks * i = 1x10-9 * (1,5+0,4)/0,4 that is 150 mm/a
Calculating with hydrus, I got 282 mm/a. What happen? Why is it so much higher?
Looking trivial, and a little bit off topic, but maybee you have an idea...

Regards
Uwe

Jirka, 12/08/2005

Perhaps the gradient across the sealing layer is larger? Is there zero pressure at the bottom of the sealing layer? Perhaps not.
J.

### ID = 482, Steady-state flow and solute transfer

stathi02, 12/12/2005

Hi my name is Stathis Diamantopoulos and i would like to ask you about the hydrus program. I want to simulate a steady state flow solute transport. Which parameters do

i have to use? For example is it necessary to use the soil parameters? I only want to use the water flux,the column length and the transport parameters. Can i do this?
Sorry for my English is not my native language
Thank you in advance!
(to help you,How can i simulate Steady state flow?)


Jirka, 12/12/2005
Stathis ,
when you simulate steady-state saturated water flow, then only two soil hydraulic parameters are relevant, i.e, the saturated water content (porosity) and the saturated hydraulic conductivity (flux). All the other parameters (residual water content, and shape parameters alpha, n, and L) are not needed and you can leave it at any values.
Jirka


stathi02us, 01/19/2006
About other options such as intial water content? What initial water content value should i use?


Jirka 01/19/2006
When you seek the steady-state solution, you should specify the initial water content as close as possible to the expected solution. Only in such case HYDRUS will find the steady-state right away. Otherwise, you will have to do time-marching, i.e., run it for a long simulation time until the solution stops changing.
J.


**ID = 483, Cd transport**

Ggghhhiii, 12/14/2005
Hi i want to make a test simulation about cd transport. Which is the most apropriate way to simulate Cd transport with the hydrus 1d model?.Thank you very much!


Jirka, 12/14/2005
Check out the following publications:
Seuntjens, P., K. Tirez, J. Šimùnek, M. Th. van Genuchten, C. Cornelis, and P. Geuzens. Aging effects on cadmium transport in undisturbed contaminated sandy soil columns. Journal of Environmental Quality, 30, 1040-1050, 2001.

Seuntjens, P., D. Mallants, J. Šimùnek, J. Patyn, and D. Jacques, Sensitivity analysis of physical and chemical properties affecting field-scale cadmium transport in a heterogeneous soil profile, J. of Hydrology, 264, 185-200, 2002.

If you want to consider more complex chemical reactions than are available in HYDRUS-1D, then you should try the HP1 (HYDRUS-1D-PHREEQC) code.
HP1: http://www.sckcen.be/hp1/

HP1 Course & Workshop Feb. 2006
http://www.sckcen.be/sckcen_en/activities/conf/conferences/hp1/index.shtml
Jirka


**ID = 484, Trouble converting projects from older versions**

Hector, 12/15/2005
Dear group,
I have to check some calculations I did with an older Hydrus Version. This time Hydrus checked that the file I wanted to open was an old one and made the conversions (Pcp_File_Version=2 at the top line) however errors appeared when trying to access the domain.dat (which stored as was an ASCII-file)
I do not understand this problem. Several times I have converted projects into the 2-version. Do I have to edit domain.dat and adjust something so that my H2D 2.008 version can read the projet?
Anybody any hint on how to handle this bug?

regards
Héctor Montenegro


Mirek, 12/15/2005
Hector,
send me the project, I'll look at it. Please use WINZIP if you can.
Mirek


Hector 12/15/2005
Hello Mirek,
very kind, thank you.
your spamblocker seems to be pretty efficient and does not accept my mail with the zipped project. Any idea how to overcome this?

Regards
Hector

<mireks@pc-progress.cz>: host relay.zoner.com[217.198.112.198] said: 554
Service unavailable; Client host [141.38.3.230] blocked using
bl.spamcop.net; Blocked - see http://www.spamcop.net/bl.shtml?141.38.3.230
(in reply to RCPT TO command)


Mirek, 12/16/2005
Hello Hector,
There is really an error in the function for reading of old FE-Mesh, I'll have to fix it. However, you can solve the problem easily: open file MESHTRIA.TXT and insert "0" at the beginning of the first line, i.e. original line was:
1030 2954 1925 133
and the new line should be:

0 1030 2954 1925 133

After that I was able to open the project and work with it.
Best regards
Mirek


Hector, 12/16/2005
Hello Mirek,
thank you for your help. Including the nought in meshtria.txt allowed to open the project, however I am not able to modify and rerun it: Error when reading from an input file dimensio.in. Then I included the "Pcp_File_Version=2"-header in dimension.in but again it did not work.
Do I have to convert everything into binary, before continuing the modifications? Any hints on how to handle this?
Regards
Héctor


Mirek, 12/16/2005
Hector,
I can modify the project here (run calculation, etc.). I use the following steps:
1/ Create a copy of your original project (WLS11_Original)
2/ Insert "0" into the MESHTRIA.TXT
3/ Open WLS11.h2d. During loading of the project you will be asked if you want to convert the project to the new format. Click "Yes". Then you will be prompted for a directory to save original files. Press "Cancel" (you already have a copy).
4/ DO NOT run module "Geometry and FE mesh editor". This module can not read the old geometry. If you want to modify geometry or FE-mesh you will have to create a new project.
5/ After previous steps I am able to run boundary conditions editor and modify input data or I can run the calculation and display results. If it doesn't work on your computer then I could send you my data or you can send me the modified project so that I could check it.

Regards
Mirek


Hector, 12/19/2005
Thank you Mirek,
in fact I wanted to modify geometry and make some mesh refinement. I have a couple of those files so I will make my mind weather it is faster to recreate the whole thing or to reinstall version 1 (if I only knew where it was...). However, I recall having transferred version 1 files into H2D ver.2 and I could modify them without any problems. What was the buzz with these projects? The non-binary mesh data?

Regards
Héctor Montenegro

Mirek, 12/19/2005

Hector,

If you want to modify geometry or FE-mesh then I'd recommend you to create new project(s) in version 2.xx. The version 1.xx is very old (1999 or earlier) and it would take some time to find and prepare the installation.

As for the error - I think that the conversion works correctly for binary FE-mesh files. I remember that we tested it and also no problems with conversion have been reported.

Regards Mirek


**ID = 485, Observation nodes**

Anafiz, 12/15/2005

Hi Jirka

I am simulating a straightforward one-dimensional soil profile in a column for flow of well and grey (laundry) waters. I put a head difference of 0.5 foot on the top of the 2-foot soil profile. I specify three observation nodes in the "Soil Profile-Graphical Etitor" for output of the pressure head and water content at each time step, but I only get one curve when I click on "Observation Points" from the "Post-Processing" menu. Shouldn't I get three curves instead of one?

Thank you.

Ahmad


Wibri, 12/15/2005

Any chance it is a scale issue and all three curves are on top of each other?


Anafiz, 12/15/2005

Hi Wibri , I don't think it is a scale issue and all three curves are on top of each other as you guessed. I change the scale but still get one curve for the there observation nodes. Also, if I view the legend, I see three items (line segments) labeled N1, N2, N3.


Anafiz, 12/15/2005

Wibri,

You might be right for the scale issue and all three curves are on top of each other. Because when I selected any of the data series for the nodes (by right clicking on the curve) and changing the smoothing (for example Quadratic B spline) I can observe off set on the curves. But you only split one curve from the other two by this method. How do you do it to split all the three curves?

Thanks

Ahmad

Wibri 12/16/2005
Anafiz,
You can see if the data is exactly the same by looking in the obs_nodes.out
If the numbers for all three nodes are exactly the same, then they are exactly the same. If not, then you should be able to separate them by playing with the range of the y-axis.

## ID = 486, Weighting in inverse modelling

Wibri 12/15/2005
Hello,
Anyone have general advice on how to weight data for inverse modeling? I have cumulative drainage and water content data. The relative uncertainty of the water content data is much higher. Is there a good rule of thumb?

Jirka, 12/15/2005
Check out the following literature:
Šimùnek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

Hopmans, J. W., J. Šimùnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Jirka

## ID = 487, Boundary nodes error

Repease, 12/19/2005
I am receiving the error message, 'Error in FE mesh! Check that you do not have individual nodes on the boundaries.'

This same message was posted by Jason, in June, but I am not clear on the responses. The node number that Hydrus is having a problem with is located in the interior of my mesh. I created the mesh with MeshGen and checked the geometry.

Maybe I am not understanding what boundary we are discussing, but I do not know how to begin to troubleshoot this problem. I would appreciate any assistance.
Sincerely,
repease

Mirek, 12/19/2005
Hi Repease,
This error is usually caused by overlapping nodes but there can be also other reasons. If you can not find where the problem is, please send me the project, I'll look at it.
Mirek


Mirek, 12/20/2005
Eric,
The problem was really in overlapping points. Definition points of your internal curves (horizontal lines inside domain) overlap other single points. Try to delete or move internal curves and you will see these points (for example delete the top left horizontal internal line). Unfortunately the consistency check function didn't find this problem (I'll have to look at it later…). You must delete these points under curves - select them with rectangle and press "delete".

Eric, I think that there would be more problems with your current FE-mesh. I'd recommend you to make some changes especially in definition of internal lines. I'll send you files with fixed geometry today or tomorrow
Regards Mirek


## ID = 488, Fracture rock

Srilert, 12/20/2005
Dear all,
Could you advice me when I use HYDRUS 2D simulate in fractured rock?
If not, which model that I can use for simulation heavy metals in fracutures rock?
Regards,
Srilert


Jirka, 12/20/2005
HYDRUS-2D does not simulate flow in fractured rock unless you can use the options with immobile-mobile water. In HYDRUS-1D we have already implemented an option that considers transient mobile-immobile water, i.e., water can flow in and out of matrix, but is immobile while in the matrix, while flow occurs only in the mobile zone (fractures). It seems to me that this option could be used to simulate flow in fractured rock. HYDRUS-2D at present does not have this option. We have, however, included it into a new release of Hydrus-2/3D, a beta versin of which should be available early next year.
Jirka


Srilert, 12/23/2005
Dear Jirka,
I would like to know about HP1, have it option mobile-immobile water like your answer in HYDRUS 1D ?
Regards,
Lertc

Jirka, 12/23/2005
Yes, PH1 has MIM model. More information on HP1 is on:
NEW: HP1 Course & Workshop Feb. 2006
http://www.sckcen.be/sckcen_en/activities/conf/conferences/hp1/index.shtml
http://www.sckcen.be/hp1/

Jirka

## ID = 489, Importing initial conditions bug?

Jkrivic, 12/29/2005
Hi!
I was running a transport simulation in a variably saturated soil profile for two days. I've assigned 240 print times because I wanted to get a detailed presentation of processes in the graphical display of results. After examining the results I noticed that the transport did not reach the required distance in the duration of simulation. I wanted to run the simulation again, starting from the end time of the first simulation. I tried to import the initial conditions (concentration and head) from the 240th print time of the first simulation but the Import Initial Conditions dialogue only allows for the input of values between 0 and 100!
Is there a way around this obstacle?
Happy holidays!
Jure

Jirka 12/29/2005
Jure,
Are you sure about the limit of 100. I have checked our code:
DDV_MinMaxUInt(pDX, m_nLayer, 0, 1000);
and it has limit of 1000 (see above). Nothing can be done about this limit, except that we would have to change this limit and recompile the entire program.

J.

Jkrivic, 12/30/2005
Jirka,
It seems that only 100 time levels of the initial simulation are recognized, although all 240 are displayed in the graphical display of results.
I'm sending you a more detailed description with screen prints via e-mail.
Jure

Jirka, 12/30/2005
Jure,
If the display part of the program recognizes 240 time levels, while the import part does not (it sees only 100), that must mean that the two projects (i.e., current into which you try to import and the one from which you try to import) have different fe-

meshes, i.e., different number of nodes. Both project must have the same number of nodes. If they do not, then indeed the suggested time would be different from your expected print time (for the 100th or 240th print time level).

Jirka

Jkrivic, 01/03/2006
Jirka,
The project is an identical copy of the previous one. Only the initial conditions are not the same (but that is the purpose of this process anyway). No mesh changes or any other changes have been applied.
The number of print times is indeed 240.
Jure

## ID = 490, Handling huge out files

Jkrivic, 12/29/2005
Hi!
Can somebody give me a hint on how the solute.out file is best handled?
In some of my transport simulations I get huge solute.out files (almost 0.5 GB!!)
Opening the Bounadry Solute Fluxes diagrams in Hydrus postprocessor literally takes hours! And even then I can not export the selected chart data. (hint for the next Hydrus upgrade: That option would really come in handy!)
On the other hand, the solute.out is so large that importing it into Excell would require breaking the original file in several tens of smaller files.
How do users usually cope with this sort of problems?
I guess I'm looking for a way to selectively pick the calculated data from the solute.out file in time steps, that would be much larger than the ones in the file, but significantly smaller than the print times.
Best wishes,
Jure

Jirka 12/29/2005
Jure,
In the version (HYDRUS-3D, that will include 2D as well) that we will release at the end of next month we will allow users to specify prints at regular intervals and after each n-time steps. With the current version, you need to write a short program that will read only each n line and delete the others.
Jirka

Jkrivic, 12/30/2005
Jirka,
Unfortunately, I'm not familiar with computer programming. I guess that such programs are widely used. Is there such a program publicly available?
Jure

Jirka, 12/30/2005

I used to use this simple FORTRAN program that rewrote the original file and used only each 10 (iPrintLines can easily be changed) line.

```
c---------------------------------------------------------
c Program shortens long files by making a selection of each n-line

      character cFileName*80,ch1*1,cText*200

      cFileName = 'LEVEL_01.DIR'
      open(10,file=cFileName, status='old')
      read(10,100) cFileName ! Read the name of the file to convert
      close (10)

      open(5,file=cFileName, status='unknown')
      write(ch1,'(i1)') 1 ! New file has "1" at the end
      iLengthPath = Len_Trim(cFileName)
      cFileName(iLengthPath:iLengthPath) = ch1
      open(6,file=cFileName, status='unknown')

      nScipLines=9 ! Number of lines to skip at the beginning of file
      nDataRows=10000000 ! Number of rows in the file
      iPrintLines=10 ! Print each iPrintLine

      do 10 i=1,nScipLines
      read(5,100) cText
      write(6,100) cText
10    continue
      kk=0
      do 11 i=1,nDataRows
      kk=kk+1
      read(5,100) cText
      if(kk.eq.iPrintLines) then
      write(6,100) cText
      kk=0
      end if
11    continue
      stop
100   format(a)
101   format(20e11.4)
      end
c---------------------------------------------------------
```

## ID = 491, Rainfall

Agua, 01/02/2006

i am simulating the metal removal behavior through a sandy-filled column, aiming to apply it on-site to treat the runoff.

1. this column is put underground above water table. but i think it should still be treated as saturated since when runoff flushing into, there is no air pore left. Am i right?
2. given an average annual rainfall,say,1200mm. how can i test the life of the column from Hydrus-1D? i know the column's adsorption capacity but couldn't find any rainfall/runoff catalog in Hydrus-1D to help. Any advice?
Thanks


Mvcallaghan, 01/05/2006
I would use real daily climate data as input to a rainfall runoff model to get an estimate of the daily runoff that might be entering your column. Then apply this runoff to your column using an atmospheric boundary condition including evaporation. You'll have to come up with a conceptual model of you bottom boundary condition e.g. free drainage, constant drainage, or constant pressure (similar to a constant water table depth).

Do a round of runs with no transport to get your water balance right (e.g. a 1% annual tolerance), then add the chemical transport. You'll have to take into account any adsorption and precipitation reactions.
Mike


Agua, 01/25/2006
thanks mike, i did it, but "numerical solution not converge", now what does that suppose to mean? where shall i try to fix?
shall i tick "inverse solution box" if i do not have experiment data?
i just feel lost and cannot communicate with it.
thanks for any help


mvcallaghan, 01/27/2006
I've seen the non-convergence problem before. Are you getting through a partial run before a crash or is it crashing at the start.
1. Problem could be with your time step, node spacing or iteration criteria. Try a time step of 1E-5 days, node spacing of 10 mm, and max iterations of say 200. Set the tension itervals to zero, that has worked for me (try this one first).
2. Problem could be with your boundary conditions. What are your lower and upper BC's set to? If you have high intensity infiltration events, and the upper bc is set to "atmospheric with runoff", then you may see instability on that boundary. Can you get convergence with lower intensity infiltration? Try setting the upper BC to "atmospheric with surface layer" of 1 m (make sure units are consitent with your model).

Mike


agua 01/30/2006
it was the latter one, stuck at the very beginning, i will change those parameters you advices. thanks

a new urgent, do you have any idea what does that prompt means?
"error reading file c:\...\...\filename\\solute1.out"
cheers


mvcallaghan, 01/30/2006
Likely, your input files are buried too deep in the file structure. There is a limit to the length of the file pathway. Beyond that limit, Hydrus can't read the filename.
I would suggest putting your input files into a directory right off your C: root. For example C:\Hydrus\....
Mike


agua 02/03/2006
mike,
have you ever come across that prompt before?
"file is too big to be displayed entirely, auto selection has been made", and then even i changed different precipitation(rainfall) data in,
no difference has shown and this prompt keeps haunting.
what shall i do to fix it?
thanks


Jirka 02/03/2006
Do not change anything. The graph can display only 6000 points. If your data file is longer, the software needs to make selection which points to show. It thus takes every second, or third, or fourth, etc, depending on how big the file is. And it tells you about it.
J.


agua 04/11/2006
"Do a round of runs with no transport to get your water balance right (e.g. a 1% annual tolerance), then add the chemical transport. You'll have to take into account any adsorption and precipitation reactions."

mike, could you elaborate how to make the water balance right?
now i am using hydrus-1D to simulate the groundwater recharge case where daily rainfall data and water table elevation data are given in detail.
i attempt to know how much water flux into water table and howe long it takes, given an rain event.
to make it easier, i first input rainfall recharge as a constant flux and see the result(how much it comes out at bottom and how long it takes), yet when i set upper boundary flux other than zero, the model says "the numerical solution has not converged!" while it keeps calculating.
what is wrong?
thanks

mvcallaghan 04/20/2006
It could be one of many problems. So I can only give some suggestions at this point.
1) make sure your minimum time step is small enough e.g. 1E-5 days
2) try a node spacing as small as < 1cm
3) turn off interpolation tables by setting upper and lower limit of tension interval to zero (in the Iteration Criteria window)
4) try using atmospheric BC with surface ponding instead of surface runoff
5) try Van G parameters with 2cm Air Entry Value if using low hydraulic conductivity soils
6) for estimating recharge, you could try using a fixed pressure head boundary condition equal to the height of water above your bottom boundary

That's all I can think of right now.
Mike


brr7 04/21/2006
Hi all...I'm a newer HYDRUS 1D user and am very interested in these threads since I have been having a very hard time reaching convergence and was wondering if folks could give me a sense of the time it took their runs to work. By implementing Mike's wonderful set of suggestions, codes that weren't converging were converging, but for a simple case (no solutes, no heat, uniform soil conditions) it took 39 minutes to get through two days of analysis. So modeling a whole growing season seems to be very difficult, despite the fact that the model is designed to do so. I'd appreciate hearing from people what kind of run times they are getting, and also from hearing about your successful growing-season simulations (with solute transport, ET, and Precipitation events). Thanks so much!
Ben


Jirka, 04/21/2006
I have written here in the discussion forum somewhere what the optimal set of parameters usually is. If these parameters are used then the runs are very fast.

I have also written here (probably last week or the week ago) that we did for API a large set of simulations with extremely heterogeneous soil profiles (layers of sand, clay and caliche), runs 250 years long with daily values of precipitation, evaporation, and transpiration, in a very deep soil profile (30 m) and these simulations would typically take not more than few minutes. Annual simulations should take only few seconds.

Jirka


Agua, 05/04/2006
Thanks for Mike and Jirka.
i am still doing rainfall recharge into water table, and simulate the recharge amount and lag time after 30m's infiltration.
Questions:

1. i have 30 yrs daily rainfall data. what is teh max. data entry can i load to Hydrus-1d time-variable bc spreadsheet once?
2. in graph editor's initial condition, one can select "initial pressure head" thru the domain. but when i change the pressure value it causes a big difference in bottom flux(recharge) and other graphs, why? what is the reasonable initial pressure head(in meters say)?

Jirka, 05/04/2006
a) I do not remember what that maximum value is. However, the software warns you that it can not display more than a certain number of records in the GUI. If that happens you need to edit directly the input file atmosph.in (while the software is closed). There is no limit on how many records you can enter there. It is limited only by your PC storage.

2) I can not comment on that one. It can be anything, dependins on your conditions. I would start with some reasonable values, if you do not know the actual values, such as -400 cm.

Agua, 05/05/2006
what does hCritA exactly means? the coment in HELP i do not quite understand.
in my case, -100cm to -40cm work (otherwise no graph plot out at bottom flux) but have different shape, how can i interpret?
Thanks again

Jirka 05/05/2006
There is also a manual. Check out that. J.

## ID = 492, Installation

Elazar Bamberger, 01/04/2006
i don't sucess to install the sel extraction of Ver. 3003
Why?

Mirek, 01/04/2006
1. Did you download the file H1D_3001_Full.exe successfully (14.586MB)?
2. If you run H1D_3001_Full.exe (WinZIP self-extracting file) is there an error message? Does the self-extractions process fail? Be more specific.

Mirek

**ID = 493, History of changes**

Wibri,  01/04/2006
Hello,
I would like to conduct sensitivity analysis of model parameters. What is the best way of running the same simulation repeatedly, modifying the parameters each time, and keeping track of both the changes and the output?
I seem to be unable to use the "save as" function, which I thought would be the obvious choice.


Wibri, 01/04/2006
THis post was actually meant for HYDRUS 1-D. That is the program I am currently working with.


Jirka, 01/04/2006
You can use either command "Save as" to save the open project, or the command "Copy" in the Project manager to make a copy of the project. I guess the best way is to use Project Manager to create multiple projects with different input parameters and keep track of different project using the Project manager.
Jirka


Wibri, 01/04/2006
Jirka,
My main problem is that I do not want to go through the whole process of defining the problem each time I recreate a new project. When I use "Save as", I get an error. A directory gets created with the new name, but no files. I am running on XP and have administrative permission. However, the folder I have hydrus 1-D in (c:\hydrus1d) seems to not be allowing me to change the permissions. This is odd. Have you seen it? Do you think this is the problem? I change it, but then when I reopen the directory, it has reverted to read-only.


Jirka 01/04/2006
The project manager allows you to copy the existing project and to modify only those parameters you want to modify. You do not need to recreate the project again.


Wibri, 01/04/2006
Jirka,
When I open the project manager, there are no projects listed in any of the workspaces, including the ones that came with the package examples. I have Hydrus 2-D and 1-D version 2.2 installed as well as 1-D version 3.0 and am finding the same problem in all of them. However, if I make a change to the location of workspaces in version 2.2, then the same change is present if I close 2.2 and open version 3.0. I think there is some crosstalking problem between the different versions.

Jirka, 01/04/2006
This is intentional that all version of HYDRUS see all HYDRUS workspaces and projects. If you do not see the HYDRUS projects after the installation that means that you either do not have administrative rights to install the software on the PC or that you (or your PC administrator) changed the attributes of installed files. Both "File is ready for archiving" and "for fast searching" attributes must be checked. Do not change attributes of files installed with HYDRUS (or change it back to what it was after installation) and then you should be able to see the HYDRUS projects.

Jirka

Wibri, 01/04/2006
I have admin privelages on this computer. No changes were made to the files. Both boxes are checked on all folders. Still cannot see workspaces. Do you suggest removing and reinstalling everything?

Mirek, 01/05/2006
Wibri,
"Save As" + "Copy" problem - I think that the problem is in the privelege/security settings. Hydrus must be able to access its workspace directory for writing. There are many various security options that can disable Hydrus2D for that. You are the first user with this problem and therefore I think that there must be a special security settings on your computer (Hydrus2D runs without any problem on hundreds of machines with WinXP). Please contact your system administrator if you can.

Also you can try to use different directories/discs for Hydrus workspaces. If the Hydrus project manager is not able to work with Hydrus projects you can do it manually:
1/ The file with information about workspaces is in the C:\WINDOWS\HYDRUS2D.INI. It is a text file, you can edit it (but be carefull about the syntax).
2/ You can copy Hydrus projects (+ directories) to new workspaces.

Mirek

Wibri 01/06/2006
Mirek,
I am having our computer guy check into it. In the meantime your advice will give a good workaround. If I find out the exact problem, I will post it, just to warn future users.
Thanks

Wibri, 01/09/2006
Mirek and Jirka,
I solved my problem. It turns out that only "for fast archiving..." needed to be checked in the the indexing and archiving section of advanced attributes under folder

properties. If both "for fast archiving..." and "folder is ready for archiving" are checked, I cannot see the files in the workspace manager. I don't know how it got messed up in the first place, but it did.

Anyhow, now I am proceeding. Thanks for the help.

Brian


## ID = 494, Infiltration depth vs time in inverse solution

Sinishaivans, 01/04/2006

Hi,

I would like to give to Hydrus 2D my infiltration intake vs. time data as an input into inverse solution window. I want to obtain as closest iterations as possible.

So far as I saw, the available inputs are: cumulative boundary flux values, pressure head values, WC values, h(tetha), K(h), and parameters alpha, n, tetha r, tetha s, Ks.

I need the infiltration depth vs. time. Is there any way I can add this as a dummy value, or ....?

Please let me know. Thanks for any help possible!

Sinisha


Jirka, 01/04/2006

Check out the examples Disc and Crust from the Inverse workspace and their description in the manual. I think that is what you are looking for.

J.


Sinishaivans, 01/05/2006

Jirka, thanks for your answer.

I've checked the above mentioned examples. Therefore, I've concluded that my intake (infiltration) input in Y column (second column) has to be in negative centimeters.

Now, am I ok if I go (following the example) with 0 in column 3 (position) and with 1 for column 4 (weight)? Could you please confirm these presumptions?

Also, is the intake (infiltration) input going to be sufficient as an input data? Do I have to add Ks for example?

Thanks, Sinisha


Sinishaivans, 01/05/2006

Sorry Jirka:

I have done a mistake in the previous reply. I want to check with you this:

Is it ok if I go with negative values for column 2 (intake in cm), Type with zeros (column 3), Position with 3's (column 4, even though I don't still understand what the position is standing for), and Weight with 1's (column 5)? This would be then the same values as they were in the examples you were suggested to me.

If the above assumptions are correct, then the fitted values for intake don't show up (they were not computed because I've run the model the way I explained above). In this case the residual values are really big. Where am I goofing?

Also, model did not want to run on intake inputs only. I had to add Ks for example to make it work.

Thanks again, Sinisha

Jirka, 01/05/2006
Column 4 "Position" should be as follows:
1 – constant pressure head or flux boundary,
2 – seepage face,
3 – variable pressure head or flux boundary,
4 – atmospheric boundary,
5 – drains
6 – free or deep drainage boundary

Column 5 "weights" should be 1 unless you have information about measurement error.
Column 3 "Type" should be 0 for cumulative infiltration.

J.

Sinishaivans, 01/10/2006
Jirka:
Thanks again for your answer. I've followed your instructions and it worked. Only what is still puzzling me is that in the Inverse solution result I don't get any fitted values. They are all zero. Why is that? The residual values are actually the values of my input intake data. Do you have any idea why I don't get fitted values?

Thanks, Sinisha

Jirka, 01/10/2006
Sinisha
In your input:
Position = 1 (for constant head BC) for flux from the furrow.
Also lower the minimum and initial time steps to 0.01.
J.

**ID = 495, Cracking of clay**

sofia 01/05/2006
Dear all:
I'm doing a research on modeling the cracking of the clay barrier in a Subtitle-D cover using Hydrus-2D.

I assumed that the cracking initiates when all the points within the clay layer reach a volumetric water content corresponding to the plastic limit. I also assumed the PL to be roughly equivalent to 85% of the saturation volumetric water content.
From the "Graphical Display" of Hydrus-2D, by the time this water content (PL=85%saturation) is reached the pressure head in the clay ranges between -20,000 and -40,000 cm .

For those of you who are familiar with clay cracking, do you find all these assumptions reasonable? and Why?
I would also appreciate having any references on this subject.

Thank you all and wish you a happy new year.
Sofia Ghanimeh

## ID = 496, 2D laboratory data to bench mark against HYDRUS 2D

Tillotwr, 01/05/2006
Hello,
I would greatly appreciate any guidance to literature references addressing use and construction of 2D laboratory lysimeters to represent drip chemigation scenarios for developing data sets to bench mark against HYDRUS 2D simulations.

Thanks for your help,
Bill

## ID = 497, Trailings pile drainage

Zak, 01/05/2006
Hello - I'm running into some convergence problems simulating the simple drainage of a saturated tailings pile.

I'm starting with a fully saturated (pressure head in all nodes = 0) column (top boundary constant flux = 0, bottom boundary constant pressure head = 0) and allowing it to drain.
The hydraulic parameters are Ks = 6.4 m/d, alpha = 23.816 m-1, N = 1.2302, theta r = 0.0236, and theta s = 0.3769. Standard VG model.
The simulation works fine for a 1m column.
It seems like when you have a large column, you can't spatialy discretize it enough to get convergence. The tailings pile I'm simulating is a little over 300 meters tall. Can anyone suggest a fix or perhaps a more appropriate software package?
Thanks
Jirka 01/05/2006 : 23:24:25
Make one (or few) top nodes slightly unsaturated. J.

Zak, 01/12/2006
Thanks for the quick response Jirka - that seemed to help some. I'm still having problems getting the solution to converge with a 100 m column. Do you have any other suggestions?
Thanks,
Zak


Jirka, 01/13/2006 :
The column is 100 m? That's huge! You need to pay attention to the spatial discretization.
J.


Zak, 01/13/2006
I figured that was my problem. I'm discretizing more near the surface but still having convergence problems. Do you think it is even possible to simulate a column that large with the 1001 node limit?


Zak, 01/20/2006
Jirka - my problem seems to be a discretization problem. Would it be possible to modify the source code to have a higher node limit or would it become too computationaly intensive?
Thanks,
Zak


Jirka, 01/21/2006
Zak,
I do not think that you need more nodes. I have myself never used more than 200 elements. You need to, however, vary them spatially. You need to have more nodes where you expect larger gradients (at the surface, at the soil interfaces) and less nodes in other part of the profile. If this is a deep soil profile, than you can have much large elements in homogeneous parts of the column.
Jirka


## ID = 498, Increasing max number of observation nodes

Mvcallaghan, 01/05/2006
It is possible to get a compiled version of the H1d_calc executable with an increase in the dimension of the maximum number of observation nodes?

I am trying to fit a regression curve to the peaks of breakthrough curves at different depths. I am obtaining the breakthrough curves from the obervation nodes. Ten nodes (current maximum) in not enough to get a decent regression fit. I would like to try 50 or 100 nodes.

If an H1d_calc executable already exists with increased obs. nodes, then could someone let me know, and send it to me.

If I need to revise the code myself (NObsD in HYDRUS.FOR), could you answer a couple of questions? I have the Fortran source code for Hydrus6 from the USSL. Is that the latest version of the H1d_calc source code? Since the source code is made up of nine subprograms, how would I go about compiling them all together to produce one executable? Are there some DLL's that are also required to be updated?

Many thanks.


**ID = 499, Comparison between HYDRUS 2D and CXTFIT**

Kotte, 01/06/2006
Dear group,
In my diploma thesis i am modelling a tracer test in a column with Hydrus 2D. And now i want to modell it also with CXTFIT because it is an analytical solution and so i try to estimate the numerical dispersion in the Hydrus 2D-modell. In the programm CXTFIT i have selected the flux-average concentration. For this concentration type the first- and third-type inlet conditions are the same. And now my 2 questions:
1.) Is there any difference between a first- and third-type inlet condition in Hydrus 2D what would be importaint for a comparison between these two programms?
2.) To integrate the CXTFIT breakthrough curve i had to divide it by the discharge. Is this okay?
You used the solution from Jury and Roth (1990). In CXTFIT are the water flux density (Jw) and the area (A) of the column missing. And the discharge is Jw * A.


Ntoride, 01/08/2006
Kotte:
>In the programm CXTFIT i have selected the flux-average
>concentration. For this concentration type the first- and
>third-type inlet conditions are the same.
I understand the concentration mode and the inlet condition are very confusing. Let me briefly explain these concepts.

(1) The surface boundary

The third-type is generally preferred as long as the convective surface flux is equal to the total surface flux. If the convection becomes very small, diffusion becomes dominant and the third-type condition fails to describe the inlet condition. In this case, the first-type condition would be a possible approximation and better than the third-type. However, as similar to the thermal problem, we can control only the surrounding concentration, but not the surface concentration. It is necessary to either describe the boundary layer or introduce the exchange rate equation for the surrounding and the surface (see van Genuchten & Parker, SSSAJ, 58, 991-993, 1994).

>1.) Is there any difference between a first- and third-type inlet
>condition in Hydrus 2D what would be importaint for a comparison

>between these two programms?

These explanations can apply not for analytical solutions, but for the numerical solutions (HYDRUS). I would recommend to use the third-type inlet concentration in HYDRUS unless the water flux is very small.

(2) flux vs. resident concentrations
There have been a lot of arguments how we can describe effluent (or discharge) concentrations. Analytical solutions are only available for a semi-infinite system. The flux concentration, $c_f = c_r - D/v\ d\ c_r/dz$, is a possible approximation for the semi-infinte solutions. When we apply the transformation to the resident concentration for the third-type conditions, the results are identical to the solutions for the first-type condition for the boundary value problem (van Genuchten & Parker, SSSAJ, 48, 703-706, 1984), but not for the initial value problem (Toride et al., SSSAJ 57, 1406-1409. 1993). This is a very interesting property, however, I notice that this leads to most of misunderstanding for the first-type inlet condition.

On the other hand, in the case of the numerical calculation, the zero concentration gradient condition is generally used for the outlet (Paralange et al. Soil Sci. 153, 165-171, 1992). The solution subject to the zero-gradient agrees well with the analytical flux concentration for the semi-infinite system for the third-type inlet condition expect for the dispersion dominant condition, in other words, very small Peclet number $(P=vL/D)$

>And now my 2 questions:
>
>2.) To integrate the CXTFIT breakthrough curve i had to divide it by
>the discharge. Is this okay?

I am sorry I do not understand your question.

>You used the solution from Jury and Roth (1990). In CXTFIT are the
>water flux density (Jw) and the area (A) of the column missing. And
>the discharge is Jw * A.

The transport equation is for the unit area.

If any further questions, please do not hesitate to ask me. This is very important for other CXTFIT and HYDRUS users.

Nobuo

Ntoride, 01/13/2006
Dear Martin Kotte:
Martin explained his question to me regarding the solution for the dirac delta input. Since his question may be important for other uses as well, I make my comments in the forum.
He found the solution is different between Bill Jury's Transfer function formulation and the CXTFIT solution. For a narrow pulse input of mass $M\_0$, the solution is given

by Eq. (7.21) of Jury's soil physics 6th book. The definition of $M\_0$ is Eqs. (2.15)&(2.16) of the transfer function book (Jury and Roth, 1990):

C_f(0,t)=M_0/J_w *delta(t).

On the other hand, I defined the total mass added to a unit area of the soil liquid phase, m_b, for the surface bounday (Eq.(2.16b) in the CXTFIT manual):

C_f(0,t)=m_b/v *delta(t).

This difference comes from the definition of the transport equation. In CXTFIT, the transport equation is for the solution phase, whereas Jury's formulation is in principle for the unit area of the soil. Therefore,

M_0 = = m_b * theta

Similar difference can be found for the total concentration. Jury defined the total concentration as the total mass per unit volume of soil (Eq.(7.2) in the soil physics book). In CXTFIT, I defined the total concentration for the solution phase as the total mass per unit volume of solution (Eq.(3.12) in the CXTFIT manual). For nonreactive solutes in terms of the mobile-immobile model (MIM), c_t is given by

theta*c_t = theta_m * c_m plus theta_im * c_im

In Jury's formulation, c_t for the MIM becomes (Eq.(3.15) of Jury and Roth, 1990)

c_t = theta_m * c_m plus theta_im * c_im

Nobuo

**ID = 500, Constant pressure head and variable surface**

Wibri, 01/06/2006
Can anyone suggest a way to have a period of constant pressure head at the surface, then allowing that to infiltrate and go to atmospheric conditions?

Jirka, 01/06/2006
If you use atmospheric BC, then you can specify that there is a water layer initially at the soil surface (initial pressure head in the top node) and that this water layer can infiltrate into the soil profile. Atmospheric BC can add (precipitation) or subtract (evaporation) from this water layer. You need to use option of Atmospher BC with water layer.
J.

Wibri, 01/08/2006

Hi Jirka,

What if I wanted to maintain the ponded head for some time and then allow it to infiltrate? Is this doable?

Another question I have is about NaN and NaPD

I've seen both when running Inverse models. The NaPD comes out like this: 'When adding a new layer, error "dimension in NaPD is exceeded, Stop!"'

Jirka, 01/08/2006

A) It is doable and it is as I describe in my previous email.

b) There is a limit on how many parameters you can optimize (NaPD). The reasonable number is between 3 and 5. Hydrus allows 15, but that is certainly unreasonable number to reach.

Jirka

Wibri, 01/09/2006

Jirka,

I am sorry. I don't think I understand your previous answer completely.

My surface boundary conditions are such:

For 4 hours I maintain a constant 4 cm head. Then I allowed that to drain and run for several weeks with variable precipitation.

How would I maintain the constant head for 4 hours using the atmospheric boundary condition? Would I not need to konw the exact infiltration rate and supply enough water to accomodate the infiltration plus the 4 cm?

Jirka, 01/09/2006

That's correct. You would need to know an exact infiltration rate to keep the constant water level, otherwise, the code would infiltrate that water. You can obviously easily get that by running it first with a constant head, and then use the calculated fluxes as BC. Alternatively you can run it first with only constant head, then import results as initial condition, and continue the run with atmospheric BC.

J.

**ID = 501, Convergence problems**

Angelo, 01/09/2006

Hello,

I have some problems with the convergence of a water flow simulation with Hydrus 1d.

I tried to do an inverse analysis for the determination of the parameters alpha, n, l.

In a column of 6 cm, I imposed a constant flux = 0 as lower boundary condition and a time variable flux at the top boundary.

I used as initial conditions a hydrostatic distribution of pressure heads varying from 0 (top) to 6 cm (bottom).

I imposed as data for inverse solution some values of pressure head in two observation points.

These data and the time variable flux at the top boundary were measured during some laboratory tests.

When I execute the calculation, at each time step there is a writing "the numerical solution has not converged!".

Maybe I'm making some mistakes in defining the problem. Can you help me?

Jirka, 01/09/2006

a) You can not start with fully saturated column and impose flux. Thus start with slightly unsaturated column (at least at the top).

b) Your boundary conditions can potentially be unrealistic. If you have zero flux at the bottom and column close to saturation, then you can obviously prescribe only evaporation at the soil surface (there is no space available for infiltration).

c) If you try to simulate evaporation experiment, then look at the example Evapor in the Inverse workspace, that simulate that experiment. Details about it, you can find either in the manual or in (Šimùnek, J., O. Wendroth, and M. Th. van Genuchten, A parameter estimation analysis of the evaporation method for determining soil hydraulic properties, Soil Sci. Soc. Am. J., 62(4), 894-905, 1998. )

Jirka

Angelo, 01/09/2006

Hi,

I'm sorry, I didn't specify I used a time variable negative flux at the top boundary.

Aniway now I'm trying with an Atmospheric BC and unsaturated initial conditions and the analysis converges.

Thank you very much,

Angelo

Agua, 05/12/2006

hi,

i use deep drainage lower bc to model rainfall recharge thru 45m column.

firstly i set the initial head as -100cm uniformly, it works but i think it only fit free drainage case, now i am in deep drainage, so at least the bottom h>0, but after i changed initial h, there is a convergence problem, it came out half way during model run. so where may be the problem:

1. input daily rainfall at some point may vary too much, so it cannot converge, possible? so i increased time tolerence(up to 0.1) and pressure tolerence value(up to 10) then run, it went further but still failed to converge later.

2. will A,B value in deep drainage equation affect it?

3. my domain is 45m below GL, and initial water table is 38m bGL, so i set pressure head as: from 0m to 38m, h gradually reduce from -1m to 0m, then from 38 to 45m bGL (bottom), h increase from 0m to 7m. is that sensiable?

4. GWL0L=0m or 45m?

5. min time step i tried: 1e-5, 1e-6, 1e-7. none converges.

6. and i already increased node number to near 1000, and at the beginning and end denser than elsewhere. still not converge.
any advice on that problem?


**ID = 503, Tortuosity function**

mille989, 01/09/2006
I have a question about the tortuosity function that is included in HYRDUS-2D. In the User Manual the tortuosity function is listed as that of Millington and Quirk (1961) in this form
(air-filled porosity)^(7/3) / (total porosity)^2
As I look through other literature it seems most others list the Millington and Quirk tortuosity function as
(air-filled porosity)^(10/3) / (total porosity)^2
This results in very different tortousity values and very different effective diffusion coefficients. I was wondering which of these formulations is actually correct or if you could clarify for me why you feel yours is correct.

Thank you,
Nate


Jirka, 01/09/2006
Note that in the dispersion coefficient, the tortuosity factor is multiplied by water content (air content). That will bring it to the power of 10/3 also.
I have made a search of literature some time 15 years ago and you can indeed find references to both exponents.
J.


**ID = 504, Two soil conditions**

Zaydoun, 01/09/2006
Hi; I am using Hydrus 1d to simulate 1-D sand column experiment in which I am running a 10 cm of constant head of water (for 5 min). Then letting the column drain for another 495 min (total 500min). This already done by adding extra line in the time conditions. I need to know how can I assign both values for drainage and infiltration soil parameters to run the experiment. The soil parameters for the drainage are different than those of the infiltration.
Thank you for your help.
Zaydoun


**ID = 505, ZBRENT**

Wibri, 01/10/2006
Hello,
What does the "Root must be bracketed by ZBRENT" mean?

Jirka, 01/10/2006
That has something to do with dual-porosity of Wolfgang Durner. I'm using ZBRENT routine from Numerical recepies to calculate Pressure head from saturation.
J.


Wibri, 01/13/2006
What might cause that error to arise?


## ID = 506, Solute reaction parameters

Srilert, 01/13/2006
I read from HELP but still confuse of what different between these 3 parameters KS, nu and beta
Ks Adsorption isotherm coefficient, ks [M-1L3]
Nu Adsorption isotherm coefficient, n [M-1L3]
Beta Adsorption isotherm exponent, b [-]
And could anyone tell me where i can value of them of heavy metals release in soils?
Thanks in advances
Srilert


Valliyappant, 01/13/2006
Hi Srilert,
The 3 parameters Kd, Nu and beta are the key parameters which deals with solid-liquid or sediment-water or soil-water interactions. These solid-liquid interactions are explained by adsorption isotherms.

There are three different adsorption isotherms which hydrus2d deals with. They are linear, langmuir and freundlich isotherm. The below three parameters decides which isotherm you want to use.

Ks Adsorption isotherm coefficient, ks [M-1L3]
Nu Adsorption isotherm coefficient, n [M-1L3]
Beta Adsorption isotherm exponent, b [-]

For example,
Setting a value for Kd or Ks, Beta and Nu will result in langmuir isotherm.
Hope this helps.


Srilert,  01/18/2006
Dear all,
In Langmuir sorption isotherm
c*=(abC)/(1+aC)
a: an adsorption constant related to the binding energy(L/mg)

b: the max. amount of solute that can be absorbed by the solid (mg/kg)
I am not sure a,b and C mean ? or compare which parameter in HYDRUS 2D (Nu, beta and KS) could anyone tell me this problems?
And I want to know about these values of heavy metals for any soils, How should i find these values, any references ?
Looking forward for your answers. Thanks in advances.
Srilert


Jirka, 01/18/2006 : 17:03:24
Lert,
You formulate Langmuir sorption isotherm as follows
$c*=(abC)/(1+aC)$
In Hydrus it is formulated as follows:
$c*=(K\_d*C)/(1+Nu*C)$
thus clearly $Nu=a$, and $K\_d=a*b$

Beta is not used here (is equal=1), only in the Frendlich isotherm.
All these coefficients depend on soil and contaminant.


Srilert, 01/19/2006 : 02:53:16
Thank you so much Jirka.
I have another qurstion. Due to I have to try run in HYDUS 2d for example heavy metals in sand. How can I find Nu and Kd from any references or only do batch experiments. Could u have any suggestion ?
Thanks in advances.
Srilert


Srilert 01/23/2006 : 06:57:37
Dear all,
I have another qurstion. Due to I have to try run in HYDUS 2d for example heavy metals in sand. How can I find Nu and Kd from any references or only do batch experiments. Could u have any suggestion ?
Thanks in advances.
Srilert
Srilert, 01/24/2006 : 06:03:32
Dear Jirka,
I still confuse about unit of Kd and Nu when i want to input in Hydrus 2d. If i use Nu = 833.33 mg/g and Kd = 0.222 g/mg, is ot correct? or i should use Nu = 0.833 mg/kg and 0.222 g/mg.
Could u pls. explains me more ? any unit should be concerned ?
Thanks in advances.
Srilert


Jirka, 01/24/2006 : 06:17:22
Since Kd multiplies bulk density it must be in reverse units of bulk density to produce dimensionless number.

Since nu multiplies concentration, it must be in reverse units of concentration to produce dimensionless number.
Thus both your units were incorect.
J.


Srilert 01/24/2006 : 07:30:36
Dear Jirka,
Thank you so much, now i revise my parameter Kd is 0.221 L/g and Nu is 0.000266 l/mg.(Sorry for my mistake) But in my simulation I use bulk density of sand equals 1.5 cm^3/g and concentration in unit mg/l . In these cases, Should i convert kd from 0.221 L/g to 221 cm^3/g only . Is it correct?
And for Freundlich sorption isotherm, y(mg/g)=0.9521x(mg/l)-1.3128, that means Ks=e^(-1.3128)=0.266 and beta = 1/0.9521=1.05 . I am not sure that i understand clearly, is it correct.
Sorry for waste your time but i think i nearly understand in this step. Thanks in advances for your kindness.
Srilert Chotpantarat


Srilert, 01/26/2006 : 02:53:21
Dear Jirka,
Thank you so much, now i revise my parameter Kd is 0.221 L/g and Nu is 0.000266 l/mg.(Sorry for my mistake) But in my simulation I use bulk density of sand equals 1.5 cm^3/g and concentration in unit mg/l . In these cases, Should i convert kd from 0.221 L/g to 221 cm^3/g only . Is it correct?
And for Freundlich sorption isotherm, y(mg/g)=0.9521x(mg/l)-1.3128, that means Ks=e^(-1.3128)=0.266 and beta = 1/0.9521=1.05 . I am not sure that i understand clearly, is it correct.
Sorry for waste your time but i think i nearly understand in this step. Thanks in advances for your kindness.
Srilert Chotpantarat


Jirka, 01/26/2006 : 17:13:08
If you use bulk density in g/cm^3 than kd must be in cm^3/g.


Julie, 01/26/2006 : 20:51:54
Srilert : I am also a PhD student, the purpose of my work is to quantify and simulate water and KNO3 transfers in an andisol under tropical conditions.
3 years ago, I chose HYDRUS 2D for my modeling and in one way, I understand your multiple questions. I recommend you to buy and attentively read the manual, to print all the forum topics you are interested in, to understand and reproduce model examples (it is a very good exercice, especially for units!).
If you want to contact me, you can. I did some batch experiment in order to evaluate Langmuir parameters.
Happy new year to you, Jirka and the members of the Hydrus 2D forum !
Julie

Julie 02/03/2006 : 15:57:54
Hello,
You could find an interesting litterature survey for metal partition coefficient at
http://www.epa.gov/athens/publications/reports/Ambrose%20600%20R%2005%2007
4%20Partition%20Coefficients.pdf
Have a pretty day
Julie


Srilert, 02/04/2006 : 03:43:13
Dear Julie,
Thank you. i got it alraedy. i think it very usefulness.
Srilert


## ID = 507, Modelling adsorption

Mary, 01/13/2006 : 22:52:45
I am trying to simulate solute transport (with adsorption) using the Freundlich isotherm in 1-D column. Can you tell me where in what output file can i find the final amount of the adsorbed solute after infiltration? The manual says that the term: ConVolIM should indicate amount of the solute adsorbed, but this term appears only when kinetic adsorption or IMM is chosen. These conditions are not applicable to my case.

Thanks for the help in advance.


Jirka, 01/13/2006 : 23:35:35
When sorption is instantaneous, the code reports only liquid concentrations since the sorbed concentration can be calculated directly using the Freunlich (or other used) isotherm and the liquid concentration.

The amount of solute (in both liquid and solid phase) is printed for different layers into the Balance.out file.
Jirka
Mary, 01/15/2006 : 18:54:29
1)So, by plugging the cMean (reported by the balnace output file) in the Freundlich isotherm we can calculate the amount of solute adsorbed to the soil? so the HYDRUS reports only (c)with time but not the (s) concentrations from the nonequilibrium transport solution?
2)Does HYDRUS take into account the initial concentration of the solute or type of adsorption (like metals differ from nitrogen)taking place?
I just want to make sure that my calculations and my interpretation of the results are right.
Thank you.


## ID = 508, Slope of water table

Srilert 01/19/2006 : 09:08:02

Dear all,

I am wondering when i want to set slope of water table in flow domain. My flow domain divided into 2 parts. 1 st part is unsaturated zone and 2 nd part is saturated zone (i want to set this part having water table with mild slope) How can I do this ?

Thank in advances for your answer.

Srilert Chotpantarat


Jkrivic, 01/19/2006 : 09:24:56

Hi Srilert,

Perhaps setting two parallel vertical constant pressure boundaries with different pressure values (both defined as "equilibrium from the lowest point") would do the trick. Bottom boundary would have to be a no flow boundary in that case.

Jure


Srilert, 01/20/2006 : 04:26:37

Thanks for your answer. I try do that already, but it can run but it 's not represent water table. However, yesterday i try to set the initial condition as pressure head and set it having slope -0.001 and then set RHS bc. as constant head and LHS BC as seepage. the results seen water table as a slope that i set. But i am not sure that i done. it's correct. Could anyone suggest or confirm me pls.

Thank in advances.

Jkrivic, 01/20/2006 : 07:34:23

I think the two constant pressure BCs should work. I remember doing it once. Did you apply slightly different positive pressures to the vertical boundaries in order to represent the gradient and the water table at the desired level?

I'm affraid that you are forcing the model a bit too much with your latest approach. If you do not have an actual seepage face, than I don't think you should use it in the model. Although, you might get a satisfactory solution if you are lucky... Check the INflux through the constant head and OUTflux thrugh the seepage face BCs, and compare them with the actual flux through the domain.


Srilert, 01/27/2006 : 10:15:19

Dear Jirka and Jkrivic and all

Thank you Jkrivic for your advice, it should be constant pressure both side, it's more realistic in gw flow. But i have other question about i want to set initial condition like a gradient groundwater flow about -0.01. I mean water table lower ground surface about 2.00 metre. At first time I run(find steady state solution in a single step ny unselecting"water flowing") for finding steady state, the program show that "No steady state". And after that i try to run for long time about 100 days, I find water content at any point is constant, and i found that pressure head at groundwater surface is - 100. why it 's so high minus vaules. Now I try to set this condition as a initial condition for solute tranport from ground surface. Is it correct ? I don't know?

Could anyone tell me?

Thank you so much .

Srilert

Jirka, 01/27/2006 : 18:58:09

You need to specify your initial conditions (pressure head) so that it has the required slope. Then you need to fix the pressure on sides of the domain using boundary conditions (constant heads).

You can not certainly have negative pressure at groundwater table. By definition, groundwater table is where the pressure head is zero.

I think that you should run first our tutorials. Mainly the one with the transect with the stream. In that we practice all this.

Jirka


## ID = 509, CXTFIT (stanmod)

stathi02us, 01/19/2006 : 15:00:38

Hi my name is Stathis Diamantopoulos.

I try to simulate solute transport with Cxtfit.

I want to use the non-physical equilibrium option.

I want to test the different f values. (mobile-immombile fraction)

I have some parameters from a previous experiment (steady state). This values are:

Z=50cm (Column length)

q=5cm/h

D=2cm^2/h

a=omega=0,1h-1

and f=beta==0.826.First of all, i have breakthrough curves in the bottom of the column.I want to ask about the column length and how can i simulate this number?

Also i use dimensionless values for both time and length.So should i change my values as the table in the cxtfit manual? For 3example omega=a*l/v*$\Theta$?

thank you very much and sorry for my english is not my native language


stathi02us, 01/21/2006 : 11:31:44

Can anyone please help? I read the manual and i can understand


stathi02us, 01/21/2006 : 15:48:44

I solve some of the previus problems but i still have one!For your help I try to simulate the first curve (Exp 1) from P.Nkedi-Kizza et al (1983)(water Resources) and i want to take the BTC from a 5cm long column. My input data are:

q=0.254 cm/h

$\Theta$=0.579 cm3/cm3

Pulse T1 Pore volume

P=4.3

Then i calculate the following values

D=0,51

v=q/$\Theta$=0,4386

I choose dimensionless both Time and length and I use the flux concentration and for characteristic legnth i put 5 (Collumn length)

I also choose a Pulse input at aplication time condition with t=0.376

I only want the BTC at the bottom of the column (5cm) but the curve is not so good. I noticed that when i choose the initial value for output position 1.2 then the curve is very good.

I only want to know if i have a mistake in this example and how the outpouts (Positions) works

Thank you very much!

Sorry for my English is not my native language!

Please answer i am in great need

Ntoride, 01/22/2006 : 17:04:04

I am not sure whether I understand your questions properly. Please
look at the example projects for Fig. 7.9 in the manual.
>Also i use dimensionless values for both time and length.So should i
>change my values as the table in the cxtfit manual?
Yes, CXTFIT uses non-dimensional parameters for the nonequilibrium
models.
>I choose dimensionless both Time and length and I use the flux
>concentration and for characteristic legnth i put 5 (Collumn length)
>I also choose a Pulse input at aplication time condition with t=0.376
Is this application time also dimensionless?
>I only want the BTC at the bottom of the column (5cm) but the curve is
>not so good. I noticed that when i choose the initial value for output
>position 1.2 then the curve is very good.
What you are saying is probably you could not fit the BTC when you fixed
v and z. It is generally difficult to know the exact saturated water
content, theta, because of entrapped air. You will overestimate theta
when entrapped air is in the soil column.

If any further questions, please do not hesitate to ask me.
Nobuo

stathi02us 01/22/2006 : 20:58:23

Dear ntoride

First, i want you to know that i am tryibg to simulate the (Exp 1) from P.Nkedi-Kizza et al (1983)(water Resources).in this experiment the column is 5cm long.This example has the folowing input values.

q=0.25cm/gr (From this value i calculate $v=q/\Theta$ and i put this value in the model)

Water content 0.579

P=4.3 (dimensionless)

From this value i calculate the D value and i put it also in the model

R=1.1

Pulse T1 Pore Volume :0.376 (also Dimensionless)

I use in my example both dimensionless for length and time.

In the concentration window i am prompted about a characterisic length.So i put the value 5 (Column length). And i want to have the BTC in the bottom of the column

(5cm). What value should i put for the the initial value for output position? For number of output positions i pou 1 (i want only one BTC)

stathi02us, 01/22/2006 : 21:00:16
I forgot to tell you that i am simulating a Determenistic equilibrium model.

Ntoride, 01/23/2006 : 06:43:12
Stathis:
If you choose nondimensional time and position, input values are also dimensionless. If the characteristic length is the column length (L=5cm), as you stated, the nondimensional outlet position is equal to 1 (=5/L).
Nobuo

stathi02us, 01/23/2006 : 10:27:20
Dear ntoride.
Thank you very much for your answers. You said that all values must be dimensionless. So how should i change the v value? We have q (cm/h). To calculate v i divide q with È. Is this correct? (È are cm3/cm3). Sorry for bothering you!
Thank you in advance!

Ntoride, 01/25/2006 : 00:26:19
Stathis:
v and D are dimensional regardless of the dimension of t and z. This is a necessary compromise to allow users to use either dimensional or dimensionless t and z. As mentioned in my previous mail, since it is generally difficult to know theta accurately, it may worth while to estimate v and D together using dimensional t. Then you can evaluate theta based on the water flux and estimated v. Please see p.67 in the manual.
Nobuo

stathi02us, 01/26/2006 : 11:54:28
Dear ntoride
Thank you very much for your annswers. They helped me a lot.

**ID = 510, Fract and thlmob**

Srilert, 01/20/2006 : 05:22:54
Dear all,
How could I find solute transport parameters in Hydrus 2d ?
1.Fract.
2.Thlmob.
I read from help but I cannot understand.
Could anyone explians me more clearly ?
Thanks in advances.

Srilert

jkrivic 01/20/2006 : 08:14:26
Did you also try checking the documentation besides the help file? You might find your answer there. The documentation can be downloaded from this web site.
Although, if one doesn't wants to go into technical details and wants to use the Hydrus software as a black box, then the documentation is too technical and the help file is not explanatory enough. There is a serious lack of thorough but still simple Hydrus software manual for that type of users.
One could buy a Hydrus book, that is being advertised on this site. Sadly, it almost does not cover the solute transport at all. There is no extensive manual on simulating solute transport with Hydrus.

Valliyappant, 01/20/2006 : 18:31:29
Srilert,
Solute transport:
Fract - Chemical non-equilibrium
Thlmob - Physical non-equilibrium
Equilibrium represents time independent sorption or reaction between soil and water. Non-equilibrium represents time dependent sorption. The time dependent reactions are the case in real world situations.

The reason for using the terms Fract and Thlmob is to get the simulation results close to the reality. The term reality mainly depends on the modeller and can represent any of the following

1. Field tests
2. Laboratory tests
3. Inversion
4. Case studies
5. Literature studies, etc

These two parameters help the modeller to get reasonable results with acceptable percentage error.

The terms Fract and Thlmob help the modeller to represent the following sorption conditions which are equilibrium sorption, non- equilibrium sorption, mobile water and immobile water. These four conditions decides the solute concentration in the domain along with Kd, Co, Nu, Beta, Alpha, etc

Setting a value (must be less than 1) for Fract will decide the fraction of equilibrium sorption sites and the non- equilibrium sorption sites.

Setting a value (must be less than 1) for Thlmob will decide the mobile and immobile water content.

You should also use the term Alpha along with nonequilibrium condition because alpha decides the reaction rate.

Alpha - First-order rate transfer coefficient for nonequilibrium sorption when chemical nonequilibrium is simulated, or for exchange between mobile and immobile liquid regions when physical nonequilibrium is simulated, w [T-1]

PLEASE NOTE: WAIT FOR JIRKA'S REPLY

Srilert, 01/23/2006 : 05:54:27
Dear Mr V.Thinnappan,
I read your reply already but I would like to ask again what is the definition of Alpha? When i go to manual, I think it 's the flow parameter right? how does it related to solute transport ? Could u pls. explain me more ?
Thank you for ur answer about Fract and Thlmob that make me more clearly.
Looking forward for your answer.
Best regards,
Srilert

Jirka, 01/23/2006 : 17:25:36
The help of HYDRUS says about Alpha in Solute Reaction Dialog Window:
Alpha First-order rate transfer coefficient for nonequilibrium sorption when chemical nonequilibrium is simulated, or for exchange between mobile and immobile liquid regions when physical nonequilibrium is simulated, symbol(w) [T-1]
Srilert, 01/24/2006 : 03:57:27
Dear Mr V.Thinnappan and Jirka,
I have other questions. How can i find the vaule of "Alpha", "Thlmob" and "Fract" in practical ? Could you pls. tell me in details or refer to some text book that i can go to details for designing my dissertation ?
Thanks in advances.
Srilert

Jirka, 01/24/2006 : 06:13:10
You need to run some transport experiments (columns in the lab) and then fit analytical (STANMOD) or numerical (HYDRUS) solutions of governing equations.

Jirka

**ID = 511, Solute boundary condition**

Srilert, 01/20/2006 : 09:40:37
Dear all,
In my flow domain, I would like to simulate the contaminants from surface as flow boundary condition at surface is "No flux" like a no rainfall situation. In this case, how should i set the solute boundary condition if i want to simulate contaminants transport from surface continuously for 24 hours with constant concentration ?
Thanks in advances.
Srilert

Valliyappant, 01/20/2006 : 19:12:17
Srilert
The following steps are to be followed based on the assumption that solute transport is based on diffusion.

1. Boundary conditions editor
2. Condition - Drop down menu
3. Boundary condition - Solute transport
4. Choose - First type
5. Select the surface with the mouse and set value as 1

Jirka, 01/21/2006 : 00:05:36
You can not specify solute transport boundary condition on "No flux" boundary. "No flux" boundary is indeed no flux, and thus nothing can pass across it. HYDRUS will not allow you to specify solute transport boundary on "No flux" boundary.

**ID = 512, Solute transport**

Jason, 01/20/2006 : 18:38:35
I'm trying to simulate vertical solute transport but the model won't run and keeps giving me the "no steady-state solution found" prompt. Any thoughts? Thanks.
Jason

Valliyappant, 01/20/2006 : 19:02:15
Jason
Please try the following by changing one input at a time
1. Increase the maximum time step.
2. Play around with time discretization.
3. If you don't have any restrictions in the final time then try changing it.
If you still have problems then check all the input parameters.
Hope this helps.

Jirka, 01/21/2006 : 00:03:16
If you disable the water flow check box, Hydrus tries to find the steady-state solution for given boundary conditions. It will start with the initial conditions that you specify and then will iterate until it will find (or not) the solution. Since the success of this process depends on the specified initial conditions (how far they are from the final steady-state solution), it is important to specify reasonable initial conditions.

If HYDRUS does not find steady state solution this way, then you need to run the simulation with the water flow check box on, keep the boundary conditions constant, and run it for a long time, until the solution stops changing (compare solution at two print times or look at fluxes). Then you can import this final solution as the initial condition for the next simulation.

Jirka

**ID = 513, Observation node**

Srilert, 01/21/2006 :  05:47:10
Dear all,
In Hydrus 2D allow to input observation node upto 10 node. If I want to input more 10 node, could u suggest me how should i do this?
looking forward for your reply.
Thanks,
Srilert

valliyappant 01/21/2006 :  16:24:50
Srilert,
You can download the latest version or you can do it manually which will allow more than 10 nodes to insert.
Please refer to the following for more details
Topic search: 'Observation nodes - Insert'
Topic ID: 339

Srilert, 01/24/2006 :  04:39:46
Dear Mr V.Thinnappan,
I try to increase number NobsD in the dimensio.in and Nobs in the Boundary.in but when i open that project it cannot open and show error. After that I try to download and install the latsest version but it show error also. I am not sure that causing i didn't unstall the previous version from my computer before installing the lastest one. Could u help me pls ?
Any comments are gracefully come.
Srilert

**ID = 514, Upscaling from lab to real fields**

Srilert, 01/23/2006 :  08:27:05
Dear all,
If i simulate in lab and using HYDRUS 2d for validating data. And i want to use results from this for real situaton. How could i do this for real situation ? Anyone tell me more in this concepts.
Any suggestions and comments for these ideas.
Thank you in advances.
Srilert Chot.

**ID = 515, Inverse solution for solute transport**

Srilert, 01/23/2006 : 10:31:22
Dear all,
I read from HYDRUS 2D book about inverse solution, it find 3 parameters (Alpha, n and Ks). I am wondering about solute inverse solution, in these cases what parameters i can find and how ?
Could u explains in details, it 'll better if u have some examples for me.
Looking forward for your reply.
Best regards,
Srilert Chotpantarat


**ID = 516, Modelling drip irrigation**

ww7596, 01/23/2006 : 12:08:39
Dear Prof. Jirka,
I am PhD student in China, Now I am simulating water and solute distribution under surface drip irrigation. this is my first time to use Hydrus 2D.Could you kindly email me any example. thanks in advance.
Wei Wang
ww7596@163.com
Jirka, 01/23/2006 : 17:36:24
I have emailed you an example of the subsurface drip irrigation with fertigation that appeared in:
Gärdenäs, A., Hopmans, J. W., B. R. Hanson, and J. Šimùnek, Two-dimensional modeling of nitrate leaching for various fertigation scenarios under micro-irrigation, Agric. Water Management, 74, 219-242, 2005.

Best regards,
Jirka


ww7596, 01/24/2006 : 03:18:41
i've got the example. it should be helpful. thanks for your time.
Wei Wang


**ID = 517, Dual porosity system**

julie j, 01/23/2006 : 16:56:00
Hello,
I begin with HYDRUS 1D and I would like to use the dual-porosity model.
I don't see the differences between the 3 models proposed (physically) and what the parameters alpha2, n2 represent in these cases; are there any parameters that represent the exchanges between the mobile and immobile water?
Is there such a case in the tutorial, I didn't find it.
If you have any references...
Thank you
best regards

Julie


Jirka 01/23/2006 :  17:38:52
Julie,
I recommend that you look at the following review paper that reviews various dual-porosity models and gives several examples:
Šimùnek, J., N. J. Jarvis, M. Th. van Genuchten, and A. Gärdenäs, Review and comparison of models for describing non-equilibrium and preferential flow and transport in the vadose zone, Journal of Hydrology, 272, 14-35, 2003.
Jirka


**ID = 518, More solute transport!**

Jason, 01/23/2006 :  17:12:23
I've established boundary conditions for my simulations but just wanted to run it buy people to see if they would agree.
For the top of my water flow I put a constant flux boundary to represent daily rainfall based on the annual value. I didn't think the atm BC was necessary nor using time-variable BCs. For the bottom I used a constant head to represent the top of the water table (I'm assuming this is a negative number?)
For the top of the solute flow I used first-type because the initial concentrations were specified. I was unclear as what to use for the bottom so I just left it third-type.
This is a pretty simply problem so I used simple BCs. Any comments would be greatly appreciated. Thanks!
Jason


Jirka, 01/23/2006 :  17:45:28
Jason,
Water flow is OK.
SOlute transport - use the third type at the top and zero gradient at the bottom.
Jirka


Jason, 01/23/2006 :  17:48:56
OK but I'm not sure what zero gradient is. Thanks.
Jason


Jirka, 01/23/2006 :  17:58:36
Zero gradient means that solute leaves the column (or soil profile) by convection with flowing water. If you specify the third-type, then you actually have to specify solute flux and that is something you do not know at the bottom of the collumn, since that is the result of the solution.
Jirka

Jason 01/23/2006 : 18:04:46
OK, but I'm not sure how to specify this. I don't see the option for selecting zero gradient. I just see first-type, volatile-type etc.

Jirka, 01/23/2006 : 18:07:01
I forgot that zero gradient is explicitly only in HYDRUS-1D. In Hydrus-2D, specify third-type, and the code will authomatically assume that it is zero gradient when the water leaves the transport domain.

Jason, 01/23/2006 : 18:08:04
One more thing. In my graphical display of results, I'm not able to produce 1-d graphs so that I know what the concentrations are at the bottom boundary. Also, the solute and water flux post-processing tabs won't open. I need this info to incorporate into another model to simulate fully saturated conditions. Thanks for your help.
Jason

Jirka, 01/23/2006 : 18:15:42
I guess you did not install it correctly. During installation, the graph ocx is registered with the windows registration database. You can do it manually by dragging VCFI32.OCX in the C:\WINDOWS\SYSTEM32 on regsvr32.exe. If this does not work, then you need to reinstall.
J.

Jason, 01/23/2006 : 18:36:51
IT WORKED! Thanks.
Jason

**ID = 519, Mass and concentration**

Srilert, 01/24/2006 : 07:51:32
Dear Jirka and all,
Now i try to use solute tranport simulation. I go to see example.5 from user manual. And i am not clear how comes the unit of initial concentration "mmol/L" while "mass unit" defined as mmol. and length unit defined as "cm". why does it is "mmol/cm^3"? And q(cm/hour) =0.271, I check it doesn't use in simulation, why does it use for constant flux from the top of the column ?
Looking forward for your answer.
Srilert Chotpantarat

**ID = 520, Cross sections**

Besel, 01/25/2006 : 01:12:50
Can anybody tell me if there is any HYDRUS-2D version which can calculate fluxes across cross-sections within the transport domain.

Regards,
Benny


Jirka 01/25/2006 :  17:16:21
We are preparing a new release later next month of the HYDRUS package for twoand three-dimensional problems, that will include flow across internal lines (in 2D). HYDRUS will be a major upgrade and extension of the HYDRUS-2D/MESHGEN-2D software package. The new version has a completely new graphical environment for both two- and three-dimensional applications, and as such should provide users with much more flexibility and ease of use than the original HYDRUS-2D.


## ID = 521, Water and solute under surface drip tape

ww7596 01/26/2006 :  12:12:15
Dear all,
I plan to simulate water and chloride distribution under surface drip irrigation, the plant is cotton seedling. i am confused about how to describe the top(surface) boundary condition. as my knowlege, the surface near the drip tape should be saturated, and this part should be specfied as varyiable flux condition, the other part of top surface should be the weather conditon, and water leave soil due ot evaporation in this part. what i am wondering is when irrigation stop, the flux of saturated part is 0, and this part is not weather conditon(no evaporation?), how does hydrus deal with this?
Anybody who had measured the chloride longitudinal and transverse dispersivty (DL and DT) of any soil? or give me some clue to find the similar thing.
any suggestion? thanks in advances
Wei Wang


## ID = 522, No post-processing results for observation points

Anafiz, 01/26/2006 :  22:28:55
I am simulating water and solute transport (chloride) in a 61 cm long (a one-layer sandy loam soil) column using HYDRUS-1D. The soil in the surface has a layer of water of 15.24 cm that was not maintained constant (a failing or variable pressure head). I specified 35 head values for the "Number of Time-variable Boundary Records". I selected free drainage as lower BC. The column was saturated from the top down. I specified initial condition in the top node corresponding with the measured water level and selected atmospheric boundary condition with surface water layer (15.24 cm) and that this water layer can infiltrate into the soil profile. I did not add precipitation but I put evaporation from this water layer. Then I allowed that to drain and run for two weeks with zero precipitation and evaporation rate of 0.03 cm/minute. I assigned all needed transport parameters for chloride and run the model and my model converged but I am not getting any of the Post-processing results for the 3 defined observation points. I am only getting results when I hit Mass balance information. What is the reason you think?
Thank you.
Ahmad

Jirka 01/26/2006 : 22:34:01
I guess you did not install it correctly. During installation, the graph ocx is registered with the windows registration database. You can do it manually by dragging VCFI32.OCX in the C:\WINDOWS\SYSTEM32 on regsvr32.exe. If this does not work, then you need to reinstall.
J.


Anafiz, 01/26/2006 : 23:04:28
Thank you Jirka
That just solved it.


Kelley B, 09/28/2006 : 21:10:24
This happens to me every once in a while. If I close and reopen Hydrus, I am able to use the postprocessor.


Jirka, 09/28/2006 : 21:19:36
This happens when you use the command "Save as". Since using this command you redefine the pathway to the project and windows looses the definition of the x-y graph. To overcome this problem, you need to close HYDRUS and reopen it. We have fixed this in HYDRUS (2D/3D), but not in older versions of our programs.
Jirka


**ID = 523, Flux units in Hydrus 2D**

Vivekgalla, 01/27/2006 : 22:42:40
hello,
i would like to know the units of flux in hydrus 2d , as i am doing a comparision between hydrus and svflux for my thesis


**ID = 524, "Error reading"… what does that prompt mean?**

Agua, 01/30/2006 : 03:38:45
hi,
is anyone can help me out by explaining how to deal with that prompt?
"error reading file c:\...\...\filename\\solute1.out"
what is that mean?


Agua, 02/01/2006 : 02:25:35
plus, when i open other diagrams, it prompts as "not enough data".
what s the prob?
thanks for help

**ID = 525, The positive and negative head in Hydrus 1D**

Anafiz, 01/30/2006 : 22:22:08
Modelers,
I don't understand the positive vs negative head concept in Hydrus-1D.
If I am simulating water and solute transport (chlorine) in a 61 cm long (a one-layer sandy loam soil) column, and initially, the soil column was dry and the soil in the surface has a layer of water of 15.24 cm that was not maintained constant during infiltration (a failing or variable pressure head). So, the upper boundary was Atmospheric BC with surface layer. I put 15.24 cm for Max h at Soil Surface. But in the graphical editor, what should I put for the Initial Conditions group for Pressure head on Top value, and Bottom value? The bottom BC was free drainage.

Also, if I know the initial concentration of chlorine before dumping the solution in the top soil surface (313 mg/l) and I know three values for concentration from the leachate after dripping started from bottom surface. Then, where should I put these measured data? Should I put them in the Time variable boundary condition spread sheet (where you have data for time, precipitation, evaporation, cTop and cBot?) But I only have three data which is in number do not match the number of time records specified for falling head measurements (35 records). What should I put for records that have no concentration measurements but only have head data?
Thank you very much


ftjin 04/11/2006 : 20:21:05
Anafiz:
negative head stands for suction potential. soil that is unsaturated and has air along with water in the pores and the pressure head is negative i.e less than the atmospheric.Any book on groundwater would explain the concept to you. Once the soil is saturated, the pressure in the soil pores is same as the atmospheric and the pressure head is zero.
In your case, I would try the constant head BC first. if that does not work, I would try the constant flux.
Let me know if thsi was helpful.
Jaspreet


**ID = 526, Axisymmetric modelling in Hydrus 2D**

Vivekgalla, 02/01/2006 : 03:23:01
hello all,
please ignore my previous post about the flux units as i was confused with other software, i figured out the flux units in hydrus 2d, my question now is what is the difference between the free drainage and seepage face boundary conditions in hydrus2d. iam modelling unsat flow in a large diameter column with a constant flux on the top of the column.

Julie 02/01/2006 :  14:12:03
Hello,
Free drainage boundary condition represents a unit hydraulic gradient.
A seepage condition is a dynamic outflow boundary condition that change dynamically according to the flow conditions. HYDRUS assumes a uniform pressure head equal to zero along the saturated (active) part of the seepage face through which water seeps out.
Have a nice day,
Julie

## ID = 527, Diffusion through clay liner

Srilert, 02/02/2006 :  07:44:18
Dear all,
i would like to know how should i do when i simulate the contaminants in containment through clay liner into groundwater(mostly diffusion process). I set the boundary condition at the surface as clay liner like a "constant flux" equals hydraulic conductivity about $1 \times 10^{-7}$ m/sec. and after that add value of contaminants at that nodes. As i mentioned above, is it correct?
Could anyone tell me?
Thank you in advances.
Srilert Chotpantarat

## ID = 528, Clarification of variable and input locations

Jennifer, 02/03/2006 :  07:53:49
Hello All,
I am new to using the HYDRUS-1D software and I have a few questions and need some clarification after going through the v3.0 manual.
1) For controlling the type of adsorption isotherm used, do the empirical coefficients $k(s,k)$, $Beta(k)$ and $Eta(k)$ in equation 3.3 (page 40) correspond to the terms Kd, Beta and Nu respectively in the Solute Transport window?

2) Where do you input the retardation factor? I couldn't tell while looking at Test 3.

3) How do you specify the number of observation nodes and their location? I don't understand how the program know where Positions 0,1,2,3, etc are. It seems that you can specify the data observed at arbitrary locations before indicating the actual locations in the graphic interface. Is that right?

4) When considering the particle attachment and detachment, the parameters D_soil, Smax2, iPsi2 and AttachSolid2 appear in the FIT.OUT file as DIFG, SINKS1', SINKG1 and SINKG1'? I'd just like to confirm if I'm following the input parameters correctly.
Thanks for your help.
Jen

Jirka 02/03/2006 : 17:10:31
1) For controlling the type of adsorption isotherm used, do the empirical coefficients k(s,k), Beta(k) and Eta(k) in equation 3.3 (page 40) correspond to the terms Kd, Beta and Nu respectively in the Solute Transport window?
Yes

2) Where do you input the retardation factor? I couldn't tell while looking at Test 3.
Nowhere. The retardation factor can be calculated from sorption constant. If you use linear sorption then: R=1+ro*Kd/theta, and as you see you can enter these constants individually. For nonlinear sorption, retardation factor changes with concentration. R=1+ro/theta*ds/dc

3) How do you specify the number of observation nodes and their location? I don't understand how the program know where Positions 0,1,2,3, etc are. It seems that you can specify the data observed at arbitrary locations before indicating the actual locations in the graphic interface. Is that right?
Observation nodes and their location is entered in the graphical Profile Module ("Soil Profile" Graphical Editor"

4) When considering the particle attachment and detachment, the parameters D_soil, Smax2, iPsi2 and AttachSolid2 appear in the FIT.OUT file as DIFG, SINKS1', SINKG1 and SINKG1'? I'd just like to confirm if I'm following the input parameters correctly.
That's probably true. I did not change the print when adding this option.
Jirka


jennifer 02/06/2006 : 06:14:13
Thank you!


Jennifer, 03/01/2006 : 05:59:02
Hello again everyone,
I have a few more questions on input-output fields
1) In the file NOD_INF.OUT, there is a series of columns for each printed time - "Node Depth Head Moisture K C Flux Sink Kappa v/KsTop Temp Conc(1...NS)" - but there seems to be two extra columns data corresponding to "Conc". This seems strange because there was 1 solute in my simulation. Which column represents the effluent data and what is the extra column(s)?
2) Based on the manual, there are three equations for the dimensionless colloid retention function, Psi (equations 3.25, 3.27, 3.28). Can the user control, through HYDRUS, which function is used and how?
3) Can concentrations in the solid phase be used for the inverse solution?
Thanks again for your help.


Jirka, 03/01/2006 : 06:55:06

a) In addition to solute concentration in the liquid phase, HYDRUS reports also on sorbed concentrations at two kinetic sites.
b) Yes, you can select the model using ipsi1 and ipsi2 coefficients. See the manual. Below is text explaining various parameters from our upcoming HYDRUS3D.
c) Yes, I believe that kinetically sorbed concentrations are Type=15

When the Attachment/Detachment Model is used, then some parameters listed above are replaced with different parameters needed for the attachment/detachment model:

D_Soil Diameter of the sand grains, dc [L].
iPsi2 Type of blocking on the second sorption sites:
= 0: No blocking.
= 1: Langmuirian dynamics.
= 2: Ripening.
= 3: random sequential adsorption model.
= 4: depth dependent blocking coefficient.
iPsi1 Same for the first sorption sites.
SMax2 Parameter in the blocking function for the second sorption sites (smax for blocking options (1), (2) and (3), and b for (4)).
AttachSolid2 First-order deposition (attachment) coefficient, ka [T-1], for the second sorption sites.
DetachSolid2 First-order entrainment (detachment) coefficient, kd [T-1], for the second sorption sites.
SMax1 Parameter in the blocking function for the first sorption sites.
AttachSolid1 First-order deposition (attachment) coefficient, ka [T-1], for the first sorption sites.
DetachSolid1 First-order entrainment (detachment) coefficient, kd [T-1], for the first sorption sites.

When Filtration Theory is used to calculate the attachment coefficient, then the following parameters must be entered instead:

D_Soil Diameter of the sand grains, dc [L].
D_Virus Diameter of the particle, dp (e.g., virus, bacteria) (e.g., = 0.95 ƒÝm or 0.95e-6 m) [L].
SMax2 Parameter in the blocking function for the second sorption sites (smax for model (1)).
Stick. Eff2 Sticking efficiency, a [-], for the second sorption sites.
DetachSolid2 First-order entrainment (detachment) coefficient, kd [T-1], for the second sorption sites.
SMax1 Parameter in the blocking function for the first sorption sites.
Stick. Eff1 Sticking efficiency, a [-], for the first sorption sites.
DetachSolid1 First-order entrainment (detachment) coefficient, kd [T-1], for the first sorption sites.


Jennifer, 03/02/2006 : 08:14:18
Thank you, Jirka! That was very helpful.
I tried a few more things and now I have more questions.

1) For the graph, "Profile Information - Sorbed Concentration vs Depth", where is the raw data? The concentrations seem normalized, but I don't see where the data is. Are these derived quantites?
2) I tried running a simulation with the attachment-detachment model. When I set iPsi1 = 2, the program fails. Is there something I've done incorrectly?
3) I tried to use solid concentrations for the inverse solution (Type=15), but the program crashed after simulation. When I change the Type back to 4, program seems fine. Any ideas on what I may be doing wrong?
Thanks again for your speedy response.


Jirka, 03/02/2006 : 19:24:41
1. If you use Type=15 (kinetically sorbed solute with depth) then x=depth (must be negative number), y=kinetically sorbed concentration, Position=Print time level. Then you should see data in the graph.
2. Yoy need to use reasonable value for Smax, then it should work. iPsi=2 is for ripening. Nobody really know how to simulate this process and what model to use, and we just used a simple model to have something there. I would not use it unless you really know what that model does.
3. see 1 above.

jennifer 03/03/2006 : 06:47:56
If I didn't say it enough before, THANK YOU VERY MUCH!
I was able to get the sorbed concentration graph to show the observed data. However I didn't see any modelled data. It's probably something to do with my input parameters so I'll work with those some more.


## ID = 529, Water and solute boundary fluxes

Jason, 02/07/2006 : 19:50:42
Dear all,
I have a domain with constant flux for the top boundary and constant pressure head for the bottom boundary. When reading the data for "constant boundary flux" under "water and solute boundary fluxes" in post-processing, is it refering to the bottom boundary? Thanks.
Jason


Jirka, 02/07/2006 : 20:06:12
The code reports (actual or cumulative) flux over all boundaries with constant (both flux and head) boundary conditions. Thus in your case it summs up both fluxes through the top and bottom. If you want to get these fluxes separately, you need to use different BC on one side. You can, for example, use time variable head at the bottom (even when you keep it constant during the simulation).
Jirka


Jason 02/07/2006 : 21:00:27

Thanks Jirka. I'm having trouble setting this up though. Am I supposed to check the "time variable boundary conditions" box and specify the pressure head under hCritA? If that's the case, the model won't let me run it for 40 years (73000 days). Keeps saying the "atmospheric data file is too big". Thanks.

Jirka 02/07/2006 : 21:07:56
hCritA is the minimum allowed pressure head at the atmospheric BC. For time-variable head BC you need to use the GWL column.
Jirka

Jason, 02/08/2006 : 19:53:24
Hey Jirka. Just to clarify, if I want my bottom boundary to represent the groundwater table, I would just leave GWL = 0 in the time variable boundary conditions editor. In actuallity, the table is 5 meters below ground surface. Thanks.
Jason

Jirka, 02/08/2006 : 22:33:29
If you want to have groundwater table at depth of 5 meters, then you should extend the HYDRUS profile all the way to that depth and then specify the pressure head equal to zero there. If you do not extend the profile to that depth, then you are probably stuck with free drainage boundary condition.
Jirka

**ID = 530, Boundary conditions**

Vivekgalla, 02/08/2006 : 22:16:05
Dear all,
i have a domain with constant flux at the top boundary and free flow at the bottom boundary. the flux rate i planned is 0.003 g/min/ft2 what is the flux unitsin hydrus 2d. and do i need to keep seepage face at the bottom boundary for free flow condition. and also the Ksat of the soil iam placing in the setup is in ft/min , what is the unit in hydrus 2d for Ksat is it mt/min? i know it depends on the domain properties i initialized in the beginning of the problem setup but just to make sure whether am i making any unit conversion mistake.
thanks

Jirka 02/08/2006 : 22:31:29
All fluxes in HYDRUS are in [L/T], i.e., length per time, such as m/s. Sorry, no feet, gallons or ounces.

vivekgalla 02/09/2006 : 23:13:07
thanks for clarifying that jirka, i have a doubt, what does 'I'stands for in the water flow parameters, and the value of 'n' and Alpha depends on what parameters? as my soil is a crushed heap leach ore.

Jirka, 02/10/2006 : 01:35:37

L is the tortuosity parameter in the hydraulic conductivity function. Parameters alpha and n are shape parameters in soil hydraulic properties.
Jirka


Vivekgalla, 02/11/2006 : 00:04:40
jirka,
its not L it is I that i asked you, actually i know n and alpha are shape parameters is there any specific range for heap leach ore, i am just confused in that part.
thanks


vivek02/11/2006 : 00:19:53
It is lower case "l" which unfortunately looks like I. And indeed it is tortuosity parameter.
Jirka

## ID = 531, Boundary discretization

Jason, 02/09/2006 : 17:28:19
Does increasing by boundary discretization (i.e. increasing the numbers of colums and rows) give more accurate results for solute flow? I ask this because my original simulation produced concentrations at my observation node on the bottom boundary. However, when I doubled the amount of rows and columns, concentrations decreased significantly or were zero for some solutes. I'm trying to make this as realistic as possible to help validate another model. Thanks!

Jason


Jirka, 02/09/2006 : 17:36:10
Certainly. In general, finer FE mesh will lead to more precise results. Look, however, also at mass balances. If the mass balance is good, then you do not need to further decrease discretization and the results should be the same.
Jirka


Jason, 02/09/2006 : 18:14:44
OK. But I just don't know why my concentrations are decreasing so drastically with a finer mesh. It just seems that this is unlikely to happen. I compared the mass balances between the two and they're pretty much similar. I ran the model for 40 years and am pretty sure the solutes would reach the water table by then. Any thoughts?
Thanks,
Jason


## ID = 532, Solute concentrations

Srilert 02/10/2006 : 15:17:02

Dear all,
I simulation flow(constant flux = 0.2 cm/day) and solute transport by set 3rd type at the surface node (leangth about 10 cm.) equals 10 mg/cm^3 and i insert observation node beneath that nodes accordingly to depth (20, 30, 40,50 and 60 cm.). From results of obs. nodes, i found that in the graph of concentration is rapidly increase and having the max. point of conc. over than 10 mg/cm^3 (about 12) and after the max point it gradually decreases like a wave (fluctulation) and having constant conc. at 10 mg/cm^3. From above, i am not sure my result is correct or not and why ?
Looking forward for your answer.
Regards,
Srilert Chotpantarat

Jirka 02/10/2006 : 19:11:50
Lert,
This may be a typical example of solution oscillations. Read chapter 6.4.6. Oscillatory Behavior of the manual for more information.
Jirka

**ID = 533, Nodal fluxes**

sparks 02/15/2006 : 07:26:08
Hi,
I am trying to use the nodal recharge/discharge function to represent a river. I was hoping that you could point me in the direction of where I can find the derivation of equation 5.26 in the manual??
Thanks

Jirka 02/15/2006 : 17:43:02
Equation 5.26 of the manual.
First, I do not know how to write equations here in this discussion forum.

In HYDRUS the nodal flux is calculated as the product of the hydraulic conductivity ($K_n$) in a particular node, multiplied by the mean gradient in that node. The mean gradiend is evaluated as the sum of gradients in all elements that contain node n, divided by their number ($N_e$).

I have a derivation somwhere, but only by hand, and thus can not send you a soft copy of that.
Jirka

sparks 02/16/2006 : 02:10:21
Thanks for the help.
But at this point I am not simulating with ions or salts in the water. And therefore, are the activity coefficients of equation 5.26 ignored? Meaning that the equation will only have the numerator that adds the pressure heads of the dimensions i,j and k. I hope this makes sense!?

Jirka 02/16/2006 : 17:40:15
Now, I have no idea what you are refering to. What "activity coefficients" are you talking about. For me activity coefficients are coefficients (calculated by Davis's or Debuy-Huckle's or Pitzer's equations) that convert concentrations to activities. But I do not have anything like that in HYDRUS-2D. I'm using this concept only in the major ion chemistry module of HYDRUS-1D.
J.

sparks 02/20/2006 : 05:01:53
Equation 5.26 has the nodal flux in x direction is equal to the satuarated hydraulic conductivity at node n divided by the number of subelements, multiplied by the sum of subelements of gamma ^ x of i times pressure head in i direction plus gamma ^ x of j times pressure head in j direction plus gamma ^ x of k times pressure head in k direction plus divided by 2 times the area of a triangular element plus the dimensionless anisotropy tensor for the unsaturated hydraulic conductivity in the x/z direction.

In this equation gamma of i (when looked in the table of variables as there was no description in the immediate) is described as the activity coefficient in soil solution [L^3M^-1]. I found a description of gamma w, s or g on page 29 which says they are zero-order rate constants for the liquid.

So I am hoping that you can explain exactly what is meant by gamma in the context of equation 5.26.
Thanks

Jirka 02/20/2006 : 11:06:38
Gammas used in the first and second equations of 5.26 are defined in the third and fourth equations of 5.26. It is a product of geometric factors b and c and anisotropy coefficients.
J.

sparks 02/21/2006 : 00:43:51
SO what do gamma, b and c stnd for? In the list of variables b is the normalised root wat uptake distribution and c is the concentration at node n. So if there is no concentration being simulated is c ignored?

## ID = 534, Kd input values

Jason 02/15/2006 : 19:07:25
When inputing Kd values, should the values be log Kd and not Kd? Typically kd values are reported as log functions in the literature. I've been converting the log

values and ending up with numbers in the 100s which result in unrealistically low solute concentrations. Thanks.
Jason

Jirka 02/15/2006 : 19:33:10
No, it should not be log KD. It is indeed Kd.
J.

**ID = 535, Three questions on a simulation**

remi 02/16/2006 : 06:57:15
I'd really appreciate it if someone could help me with this problem. I am very new with simulation and I just started working with Hydrus 1-D. I was asked to simulate water flow in a pipe that is 82 cm long and was filled with dry sand from the bottom to an elevation of 60 cm. Water was irrigated from a watering can very fast to create a water layer on the top of the soil which was 15 cm high at time zero then started infiltrating down naturally. Evaporation could be neglected . I took 33 records for falling water elevation from the soil surface with time until water disappeared into the soil after 93 minutes. Now I have three questions that I kindly ask a help to solve

1-Do I need to check the box "Time Variable Boundary Conditions"?
And are the 33 records for elevation of falling water layer with time assigned here?
2- If not, then what upper boundary condition best describe my problem in the Water flow Boundary Conditions selection window? You have Constant pressure head, Constant flux, Atmospheric boundary condition with surface layer, Variable Pressure Head, Variable Pressure Head/Flux to choose from?
3- When I reach the profile editor to define the initial condition, what should I put for top value and bottom value for pressure head?
Thank you all

Jirka, 02/16/2006 : 17:30:52
I would you atmospheric BC with Water layer.
I would then specify initial condition in terms of pressure head (you need to know what pressures to specify, you know how dry the soil was).

In the top node I would specify pressure head equal to 15 cm.
The code will then calculate, how quickly this water layer infiltrate in the profile.
You can then compare this with your measured data (falling water layer). You can also use this information in the inverse problem and calibrate model against it.
Good luck
Jirka

Remi, 02/16/2006 : 22:26:02
Thank you Jirka. That did really help a lot. I run the model and it did converge. But to make sure I did the right thing, I entered in the Time Variable Boundary Condition

spread sheet (the one that has rows and tables for "Time", "Precip", "Evap"….etc, I entered the 33 values of falling pressure head I have under the "hCritA" that was collected with time. Is that right thing? It is not clear what does this description mean "hCritA is the Absolute value of the minimum allowed pressure head at the soil surface" did I do the right thing?

Thanks a lot

## ID = 536, Inverse solution

Srilert 02/20/2006 : 06:34:49
Dear all,
I have problems about input "Data for Inverse soution". what does these meaning of X Y Type position weight when i check box of solute transport? Now, I already know meaning about them when water flow only. Could anybody please to explain me more?
Looking forward for your answer.
Best regards,
Srilert
Julie  02/20/2006 : 14:08:42
Dear Srilert,

The meaning of x, y, type position and weight will depend on the information you want to include in the objective function. What kind of observation data you want to use : tensiometric data? Outflow data?
For tensiometric data: type 1, x time, y Pressure head measurements at certain observation node, position : position of the observation node, weight : 1 (if you attribute the same weight for each value).
I let you imagine what would be the meaning of these parameters for infiltration rate or cumulative infiltration.

Here is litterature to better understand what is inverse procedure (ID 486)
Šimùnek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

Hopmans, J. W., J. Šimùnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Have a nice day,
Julie

## ID = 537, Concetration osillation

ww7596, 02/21/2006 : 11:56:20
hi,

i am modelling water and solute distribution with a domain which has an atomspheric top boundary condition and free drainage bottom boundary, and two sides are no fux. the initial value of pressure and concentration are import from am drip irigation modeling.After a 1 days evaporation, the concentration near the soil surface is extremely oscillating, i changing the space and time discretization, the weighing mothed, the oscillatio is still there. Anyone has experience about this problem. Any suggestion are appreciated.

Wei Wang

ww7596 02/24/2006 : 06:48:47
I have partly solved the problem by refining the mesh near the soil surface, in fact, refining mesh is more effective way to circumvent the numerical dispersivity than other method(time step or upwind weight ect.)

Wei Wang

**ID = 538, Salinity stress model**

ww7596, 02/22/2006 : 09:11:52
hi,
when selecting the root water uptake model,one can considering effect of the salinity stress. i selected multiplicative mod as the solute stress model, then i chose cotton from the database, the threshold is 7.7 and slop is 5.2 (i guess the unit is dm/s), but what is meaning of Osm. Coeff.(conversion to pressure/osmotic)? the default value is 1. Is this coeff. the conversion of solute concentration to osmotic pressure or concentration to electric conductivity ? my length unit is cm and solute concetration unit is mmol/l, what the Coeff. should be? Thanks in advances.

Wei Wang

Julie, 02/23/2006 : 22:50:05
Hi Wei Wang,
 "Osm. Coeff" = a and is a coefficient to convert concentration into equivalent osmotic pressure head. Unit of this coefficient should be [L4M]
Concerning the threshold-slope function, maybe you could find information in this paper: Maas, E.V. 1990. Crop salt tolerance. In Agricultural Salinity Assessment and Management. ASCE Manuals and Reports on Engineering, No 71. K.K. Tanji (Ed.), pp. 262–304. ASCE, NY.

Regards,
Julie

ww7596, 02/24/2006 : 06:30:40
Hi Julie,

thanks for your advices. providen my length unit is cm and concentration unit is mmol/l, how can convert the concentration to osmotic pressure. i use the following approximation£º1000cm=-0.36dm/s=3.6mmol/l, So i got the osm. Coeff. is -1000/3.6(-277.78). Is this ok?
Regards,
Wei Wang


Julie
02/26/2006 : 15:42:26
Hi,
Thanks for you post.
Is this approximation in the paper cited in the manual (van Genuchten et al. (1987))? In this reference it should be better explained.
I am not sure that is the good way, but the Osmotic Pressure at a given temperature depends upon the molar concentration. The mathematical relationship is as follows:
Osmotic Pressure (atm.) = Molarity (mol L-1) * R * Temp. (Kelvin)
with 1 atm. = 10.33 m of water, R = 0.0821 atm.•L•mol-1•K-1, Temp. = 298 K
With that, maybe you could also find this coefficient.
In the manual, unit of "a" is an experimental coefficient [L4M-1] and here I am a little bit confused…
Maybe Rien VG or Jirka could better explain us?
Bye, bye.
Julie

Van Genuchten, M.Th., 1987. A numerical model for water and solute movement in and below the root zone. Research Report N°121, U.S. Salinity laboratory, USDA, ARS, Riverside, California.


**ID = 539, About weight**

Shency, 02/23/2006 : 16:48:51
When I use Hydrus 1-D to fit the virus breakthrough data from a saturated column filled with sand, I can't fit some data wihin the top of breakthrough data into the curve. IF I give more weight to these data, the result is very good. The data in the top of breakthrough data is very important. However, I don't know whether it is reasonable for me to give more weight to some data and less to others within one set of breakthrough data. Could you give me some suggestions? Thanks.

chongyangshen


Jirka, 02/25/2006 : 10:16:45
In general, weights should reflect the measurement error. See the discussion in:
Šimùnek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

However, I believe that it is justifiable to place more weights on points that you believe reflect the behavious or the system.
Jirka

Shency, 02/27/2006 : 15:50:56
Thank you very much.
chongyangshen

**ID = 540, Installation new versions**

Srilert, 02/24/2006 : 05:12:05
Dear all,
Dear Sir,
Today, I download Hydrus 2d ver.H2D 2101 from web to update my desktop version (2.007) that having license already. When I install already, It asked me,to input Reg key 1 and Reg key 2. my user code 1:261314611 and my user code 2 :2897928 . How should i do because many projects that I crete alredy with last version cannot run?. Please, help me. It's very important for me
Looking forward for your answer.
Regards,
Mr.Srilert

Mirek, 02/24/2006 : 07:25:26
Srilert,
starting from version 2.100 (September 2003) there is a new system of Hydrus2D licensing that requires activation codes. Only IGWMC (http://typhoon.mines.edu/software/igwmcsoft/) is authorized to give you these codes. Please contact them and they will send you activation codes (e-mail: IGWMC@mines.edu).
Mirek

Jkrivic, 02/24/2006 : 07:30:57
In the meanwhile you can install the old version again. I'm sure that the Hydrus support team will be happy to send you a copy of the original installation files if you don't have them anymore.

Jure
Mirek 02/24/2006 : 09:29:25
http://www.pc-progress.cz/Fr_Services_Hydrus_Downloads.htm
Mirek

Srilert 02/24/2006 : 11:08:55

Dear Mirek and jkrivic,
Thank you, now I 'm wait reply from IGWMC


## ID = 541, Simulate of heavy metal

Srilert, 02/24/2006 :  11:54:45
Dear all,
now, i just run Hydrus 2d for release Mn II to ground water table in 2 dimension sand tank experiment. I release Mn II solution from left hand side surface of sand having constant flow 1 cm./hr and having flow from Left to right side , gradient 0.025 . water table below the sand surface 10 cm. From results, it uses about 7-8 hr come to water surface. I thinks it look strange. I am not sure it's correct.
And another question that when heavy metals or NAPLs that having less or more than density of water which parameters in Hydrus 2D representing different density. Could u pls. explains me in this?

lert


Jirka, 02/25/2006 :  10:48:20
Public versions of HYDRUS do not consider density dependent flow and transport.
J.


danilo 05/29/2006 :  22:56:35
Dear all:
I have used and I'm getting use to Hydrus 2D to simulate solute transport, specifically Zn.
I am running some simulations and I'm not sure if I'm interpreting right the units, even though I consult the manuals.

The case is:
1) Water flow + solute transport in vertical plane;
2) Number of material and layers equal 4 (4 horizons);
3) Profile width equal 100 cm;
4) Units: length = cm; time=day; mass=mmol;
5) Fractional of instantaneous equilibrium = 0.75;
6) Freundlich isotherm; Beta=0.65 for all materials ; : Kd in cm3 g-1: material 1 = 3.33; material 2=2.51; material 3=1.3; material 4=1.28;
7) Alpha (kinetics sites) was estimated by EDTA extraction: material 1=0.0014 day-1; material 2=0.0085 day-1 ; material 3=0.0076 day-1; material 4=0.0106 day-1.
8) Boundary conditions for solute transport = third type
9) Domain definition: initial condition: a) concentration measured at field condition; mmol cm-3; b) sorbed concentration: ATTENTION ?: We measured total adsorbed Zn (S) but we enter the Sk as we understood that is equal to: Sk = (1-f) S.
10) It was used 10 years meteorological data (daily precipitation and evaporation);

The questions are:

1) We are looking for the amount of Zn transferred among layers (horizon) and out off profile in the simulated profile for 10 years.

2) We use some of parameters from balance.out and solute.out, however we would like to check if we interpret right the data:
a) We used ConVol in mmol cm-1 for Zn left over adsorbed at equilibrium in each subdivision (horizon) and used SorbVllm in mmol cm-1 left over at kinetic adsorbed, both from balance.out.
b) We converted values in mmol cm-1 to kg ha-1 as following:
number of mmol times 65.37 = transform to milligram Ex: 0.04 mmol = 2.6148 mg
after divided by 1000 000 to get value in Kg Ex: 2.6148 10-6 kg
So we have the unit in kg cm-1 Ex: 2.6148 10-6 kg cm-1
From that we assume this amount of Zn was in 100 cm2 (considering profile 100 cm wide and 1 cm thick) and extrapolated to 1 ha multiplying by 1000 000.
Ex: 2.6148 10-6 106 kg = 2.6148 kg ha-1 so
Kg of Zn ha-1 = mmol of Zn cm-1 * 65.37
c) Total Zn left over at subdivision (horizon) = kg ha-1 from ConVol + SorbVllm

3) We got Total Zn at initial time and subtract Total Zn at 10 years transport simulation and interpreted as the amount of Zn variation for each subdivision and for total profile.

4) We tried to cross check values as following: The balance of Zn in the domain calculated from SorbVllm (balance.out) is equal to CumchN from solute out. Is this right?
5) We also tried checking the following: The balance of Zn calculated from SorbVllm + ConVol is equal to ChemS1 from solute out?

Thank you very much,
Sincerely, Danilo

Jirka, 05/29/2006 : 23:08:37
Danilo,
9) The equilibrium sorbed concentration is calculated directly by the code from the initial liquid concentration (initial condition) and the sorption isotherm (for linear case, $s_e=f*Kd*c$). If you measured the total sorbed concentration ($s_T$) then the kinetically sorbed concentration that is entered as the input value should be $s_k=s_t-s_e$.

Mass balances:
b) It is correct that values are related to width*1 cm in perpendicular direction, i.e., 100*1=100 cm2
c) Total Zn (in the entire profile, or any subregion) = ConcVol+SorbVlIm (at any print time)
For the entire transport domain: Total Zn (final)-Total Zn (init) = sum of cumulative solute fluxes across boundaries
3) For the first layer: Total Zn (time) – Total Zn (init) = Cumulative Zn inflow (surface solute flux at time) + Cumulative Zn outflow (to lower layer at time).

For the second layer: Total Zn (Time) – Total Zn (init) = Cumulative Zn inflow (from above) + Cumulative Zn outflow (to lower layer). And so on.
4) SorbVlIm(time) - SorbVlIm(init)= CumChN(time)
5) Total Zn (final)-Total Zn (init) = sum of cumulative solute fluxes across boundaries (CumS1+CumS2+…CumS6)

I hope that this was helpful. It seems that you have all the calculations correct. Let me know if you have further questions.

Jirka

PS: Note that in our new update of HYDRUS-2D, that we call HYDRUS (2D/3D) one can define internal lines and then the code calculated actual and cumulative water and solute fluxes over these lines (e.g., from layer to layer). Thus in the new code these calculations would be done automatically by the code for you.

### ID = 542, Atm flux larger than precipitation

sofia 02/24/2006 :  12:39:38
Dear all:
I'm using Hydrus-2D to simulate the evapotranspiration phenomenon over a one-year cylce. At a certain point, the cumulative atmospheric flux (-118cm) becomes larger than the cumulative precipitation rate (47cm). As a result the evaporation rate becomes negative (47-118=-71). Is it possible??!!
If no, where do you think I went wrong. and If yes, how can I calculate the evaportaion rate?

Thanks
Sofia

Jirka 02/25/2006 :  10:52:58
HYDRUS adds specified precipitation and evaporation rates, and applies the result on the atmospheric boundary. The reported value of the flux across the atmospheric boundary thus reflects both precipitation and evaporation fluxes. Later versions of HYDRUS (I'm traveling in Europe and do not have an information from which version) separate information on evaporations and infiltrations in the v_mean.out and Cum_Q.out files. Check out these output files.

Jirka

sofia 03/02/2006 :  13:10:28
Dear Jirka
Thanks for answering but it seems that my question was not quite clear. The v_Mean.out and Cum_Q.out files show the atmospheric flux, which is equal to rain + evaporation. The value of the rain is always negative (into the domain) and the value of evaporation is always positive (out of the domain). For instance, for an atm flux of -42 and a rain flux of -44, the evaporation flux is +2 cm. So the absolute value of the

atm flux should always be SMALLER than or equal to the rain flux in order to have a POSITIVE value of evaporation.

In my application, the cumulative atmospheric flux is reaching a point where it becomes LARGER than the rain value. Thus the evaporation rate becomes NEGATIVE (i.e. flow into the domain)!!!!

This hapened at a time where the rain changes from 0 to 31mm/d. Could it be a numerical problem?

To note that the mass balance error does not exceed 0.4%.

any suggestions???

### ID = 543, Mass balance error

ww7596, 02/24/2006 : 17:23:48

I am simulating water and solute movement within a domain which has a top atmospheric boundary, free drainage bottom boundary and the two sides are no flux. Evaporation rate is 1-3mm/day, after 14 days simulation, the water balance error is less than 1%, but the balance error of solute is quite high( >40% ). i refined the mesh, using the upstream weighting, every time, i alway got a small water balance error(<1%) and big solute balance error. any suggestion?

thanks in advances

Wei Wang

Jirka, 02/25/2006 : 11:07:49

If the evaporation is involved, then you need a fine discretization close to the soil surface, where due to evaporation you may get relatively large solute concentrations. At least two top elements should have vertical size less then 1 cm. Make the top element about 0.5 cm and the one below 1 cm. See how that improve the solute mass balance.

Jirka

ww7596, 02/26/2006 : 09:50:47

hi Jirka,

the initial solute concentration is 1800mmol/l, and the element lenght near the surface is less than 1cm. i got an unacceptable solute balance error and concentration oscillation near the surface.I refine the mesh near the soil surface, the element length is less than 1mm. solute balance error is less than 1% and oscillation is acceptable. my simulation domain is 0.725m*1m, so i got 21833 nodes and 42865 element. and you can image it took too much time to get the result.

Another question is about root water uptake model, i check the multiplicative model and threhold mod, i selected cotton from the database. what is meaning of conversion to pressure/osmotic? is it the conversion coefficient from solute concentration to osmotic pressure? My concentration unit is mmol/l and length unit is cm, what the Osc. Ceoff. should be? i input an value of -277, is it ok?

thanks for your time,

Wei Wang


**ID = 544, Colloids**

Silvia, 02/27/2006 : 11:11:23
I would like to ask you some questions.

1_"Fract." and "Thlmob." in Solute Transport Parameter or "Variable ChPar(3,M)" and "Variable ChPar(4,M)": what are their meaning when modeling colloid transport? I would like to model attachment-detachment with site 1 and straining with site 2, as done in "Modeling Colloid Attachment, Straining, and Exclusion in Saturated Porous Media"; Environ. Sci. Technol. 2003, 37, 2242-2250; Bradford, Bettahar, Simunek, van Genuchten, Yates.

2_Filtration theory: is it possible to specify a different density of the colloidal particle (instead of 1080 Kg/m3)?

3_Water Flow Boundary Conditions: where do I have to specify the values of constant pressure head?

Thank you very much for your help,
Silvia


Jirka, 02/27/2006 : 23:11:31
1. I would recommend not to use these two parameters when you work with colloids. They are meant to be used only with either the physical nonequilibrium (mobile-immobile) or chemical nonequilibrium (two-site sorption) models. Not with attachment/detachment model.
2. No. That value is hardcoded.
3. In the initial profile of pressure heads. Since it is constant with time, it is kept at the value given in the initial condition.

Jirka
Silvia, 02/28/2006 : 14:55:39
1_ Thank you!
2_ Do you think it is possible to me to parameterise the particle density in the program?
I have little knowledge of programming, but I could ask somebody's help. In this case could you send me the source files?
Naturally, if I could manage, I will send you back the modified source files.
3_ Do I have to specify the constant values just on the first and last cell of the domain? Am I right?

Many thanks
Silvia

Jirka 02/28/2006 : 15:00:06
2_ Do you think it is possible to me to parameterise the particle density in the program?
I do not share the source code of this latest version with public. I can make the change for you and send you exe.
3_ Do I have to specify the constant values just on the first and last cell of the domain? Am I right?
Right


Silvia, 02/28/2006 : 15:38:26
I can make the change for you and send you exe.
So which is correct? (I'm not sure I have correctly understood)
1_ you send me the exe and I will be able to input the desired density;
2_ I send you the values of density and you introduce it in the exe and send it to me.
Thank you again
Silvia


Jirka, 02/28/2006 : 19:34:12
second. J.


Silvia, 03/01/2006 : 16:58:37
The value of density I need is 7 g/cm3.
Thank you very much!!
Silvia


Jirka, 03/01/2006 : 18:26:05
I have emailed you the files. J.


Silvia, 03/02/2006 : 08:37:57
Thank you very much!
Silvia


**ID = 545, Input data for straining model**

Alice, 03/01/2006 : 16:30:00
I am trying to simulate a percolation type experiment of colloids in a clay column. I intend to use straining, and the source is located somewhere inside the column. The blocking coefficient requires the coordinate of the location where the straining starts, in this case, at the interface between the source layer and the botom layer (I discretised the column in 3 layers). I cannot find in the description of the input files where I should introduce x0 (see eq. 3.28 in the manual).
Please, advise...
Thank you,
Alice Ionescu

Jirka, 03/01/2006 : 19:02:36
Alice
It is automatically set at the soil surface (since that's where it will be for most column experiments, which are typically homogeneous) and can not be changed by HYDRUS users. If you have heterogeneous media (such as Bradford et al., 2005), then the depth function needs to be initialized at each interface. I have implemented that in 2D for the work we did with Scott, but not in 1D. That would still need to be done.
Jirka

Bradford, S. A., J. Šimùnek, M. Bettahar, Yadata Tadassa, M. Th. van Genuchten, and S. R. Yates, Straining of Colloids at Textural Interfaces, Water Resources Research, 41, W10404, doi:10.1029/2004WR003675, 17 pp, 2005


**ID = 546, Const flux in axisymmetric problem**

Jpjones, 03/05/2006 : 04:38:35
Hello,
I have a question. I am trying to simulate a pumping test conducted at Borden back in 2001 for the purpose of comparing observed/simulated behavior of the capillary fringe during and shortly after the test took place.

I am using Hydrus 2D v2.102 serial number 1301. Below is a quick description of the system I have setup followed by the problem I am having. This is a very basic system and I am sure I am making bonehead mistake but I can not seem to figure it out.

Description:
axisymmetric flow option, 200m long x 9m deep (4001 nodes x 181 nodes).
for now, using the nonhysteretic version of the van Genuchten model along with the default sand values (until I get it running).
water table = 6.25m so I set the pressure head to vary from -2.75m at the top of the domain to 6.25m at the bottom.
The pump screen is assumed to be located along the bottom 3.65m of the left side of the domain. The pump rate is 40 liters/min for 627400 secs.

Problem:
I want to represent the pumping as a constant flux boundary condition. However, when I try to assign the flux rate along the bottom 3.65m of the left hand side of the grid, it will not allow me to input the rate. I choose the nodes, a text box pops up prompting me to input the flux rate (which is -9.009009E-006 m^3/s evenly distributed over 74 nodes), I input the rate, the box disappears and so do the nodes that I had highlighted. What am I doing wrong?

I repeated the above except I assigned the flux rate along the bottom nodes of the grid and it worked just fine (as far as I can tell without actually running the simulation).

Any advice will be greatly appreciated.
Jon Paul Jones

Jirka 03/05/2006 : 21:31:01
It seems to me from your description that the vertical side of your domain on the left has the zero coordinate x (or r). That means that the width associated with each node (2*PI*r*dx) is equal to zero (it is axisymetrical). Then you can not specified flux for these nodes since the nodal flux is equal to flux*width, and that is zero. You need to have some nonzero coordinate for the left side, if you want to specify flux BC.

Jirka


Franta 03/08/2006 : 18:28:59
Jon:
I have worked on something similar. I have used an axisymmetrical type of flow for an injection well simulation. I have set up a GENERAL mesh with an axis going thru the middle of the injection well (R = 7.6 cm). I have divided measured injection rate between individual nodes that represent a screened segment of the well (10 nodes on the well wall - top to bottom 4.6 m) by specifying a NODAL RECHARGE in BOUNDARY CONDITIONS EDITOR for those 10 nodes. This run has converged.


Jirka:
Here is my problem. I now wanted to add solute transport simulation into my problem. I turned on SOLUTE TRANSPORT together with WATER FLOW. When I checked SOLUTE TRANSPORT BC in BOUNDARY CONDITIONS EDITOR it showed NO FLUX solute transport boundary for my CONSTANT FLUX water flow boundary. I think that this is the reason that this run crashes every time.
What have I done wrong? How can I specify CONSTANT FLUX water flow boundary that would also allow for solute flux?

Thanks
Franta


Jirka 03/08/2006 : 18:37:59
HYDRUS specifies by default a Cauchy Solute BC on all boundaries, including constant flux or head water BC. You can change it to Dirichlet or volatile Solute BC, but it can not be changed to no flux. I need to see your files to be able to say what is happening, since I think that it is impossible.

Jirka


**ID = 547, Major HYDRUS upgrade**

Jirka 03/05/2006 : 23:40:55
Dear HYDRUS-2D users,
We have finally finished what we set out to do several years ago, i.e., to develop a completely new program (called simply HYDRUS) that includes a single graphical

environment for both two- and three-dimensional applications. As such the new HYDRUS is a major upgrade and extension of the HYDRUS-2D/MESHGEN-2D software package. The new version has a completely new graphical environment, can be used for both two- and three-dimensional applications, and should provide you with much more flexibility and ease of use than the original HYDRUS-2D.

Please visit http://www.hydrus3d.com (or http://www.pc-progress.cz/Fr_Hydrus.htm) for the most current information and status updates on HYDRUS. Documentation and the software itself (only a demo version of HYDRUS) can be downloaded from this site. You can find at this site information about new major features in computational modules and the graphical user interface, as well as information about how to upgrade from the HYDRUS-2D/MESHGEN-2D software package. HYDRUS is fully backward compatible and allows users to convert old HYDRUS-2D project into a new HYDRUS format.

Please, pay special attention to our new Tutorial section. We have created several "videos" that explain some of the new features of the new software. We have also created "videos" that use our original HYDRUS-2D tutorials, but now carried out in the new software. That should help you to get used to the new environment.

HYDRUS-2D was exclusively distributed by IGWMC. HYDRUS is again distributed by IGWMC, but additionally also directly by PC-Progress and other distributors (e.g., Yuko Toride in Japan; you can obtain contact addresses on the HYDRUS web site).

Similarly as HYDRUS-2D, we will be supporting HYDRUS (2/3D) using discussion forum, FAQ, or directly through personal contacts.
Good luck with the new software.
Jirka and Mirek


**ID = 548, Welcome**

Jirka, 03/05/2006 :  23:49:24
Dear HYDRUS (2/3D) users,
Welcome to our new discussion forum that was set up to facilitate the technical support and mutual interactions of HYDRUS (2/3D) users. This discussion forum will operate similarly as the original HYDRUS-2D discussion forum.

If you want to contribute to this discussion forum you must first registered at this site. Registration is free and very simple. We recommend you to provide your full name and name of the company/institution that purchased Hydrus/Meshgen program. Questions from fully registered members will be answered with higher priority than other questions.

Good luck with the new software.
Jirka

**ID = 549, Simulation for road pavement and filter drain**

Nursetiawan, 03/07/2006 : 18:03:24
Hi..
I am interested to use Hydrus-2D to simulate the infiltration within road pavement layers and the filter drain as a road drainage.
Does Hydrus provide material properties for this purpose?
How about the plan to solve the overland flow problems as an additional feature of Hydrus.
Thank you


Jirka, 03/07/2006 : 18:36:52
There were quite a few users (Defen Apul, Klas Hansson, IJsbrand de Haan) who used HYDRUS-2D for this purpose. You can google them and find more information from them (they all contributed to the forum here as well). I'm aware of two publications:

Apul, D., K. Gardner, T. Eighmy, E. Lindner, T. Frizzell, and R. Roberson, Probabilistic modeling of one-dimensional water movement and leaching from highway embankments containing secondary materials, Environmental Engineering Science, 22(2), 156-205, 2005.

Hansson, K., L.-Ch. Lundin, and J. Šimùnek, Modeling Water Flow Patterns in Flexible Pavements, Transportation Research Board 84th Annual Meeting, January 9-13, 2005, Washington DC. (Also accepted for publication in the 2005 series of the Transportation Research Record: Journal of the Transportation Research Board.)

In the second, we actually did consider overland flow (although overland flow is not in the publicly available version of the software, we included it for this application) on the road surface and its infiltration on the site of the road.

Jirka


Ijsbrand, 03/14/2006 : 20:17:46
Hi Jirka,
I was wondering did you incorporate the overland flow? And if you did, how did you deal with the accumulation of the water en the delayed runoff?
I used a arbitrary non- excisting layer of air to facilitate this. (Yes I am lazy so I would not have to calculate this myself ;-). )
The simulations produced proved to fit good enough with TDR measurements.
I wonder if there is any difference if you simulate a road embankment in 3-d? Maybe a challenge?

and Nursetiawan,

I can provide you the MvG parameters I derived from measurements of road materials (secondary materials like granulated concrete).
But I would use them with a bit off caution as they are not confirmed with other research yet.
With a bit of luck I will finish my dissertation regarding this topic this year. If my daily job does not interfere to much.

Greets IJsbrand


Jirka, 03/14/2006 : 20:28:21
I used kinematic equation at the top (atmospheric) BC. I will email you the paper.
Jirka


Nursetiawan, 03/15/2006 : 15:25:12
Hi IJsbrand
It will be very useful If you can giev me those informations.
Could you please let me know how to contact you to get it.

Many thanks for your great help.
Nurs


Julie, 03/15/2006 : 15:30:44
I am also interested...
Thanks.


**ID = 550, CXTFIT decay**

Tobias 03/07/2006 : 19:59:57
Hello,
I want to use the two region nonequilibrium model of CXTFIT for solute transport with decay only in the liquid phase (inverse problem).

The result of the inverse calibrating procedure is, that I get one dimensionless decay coefficient, but in table 3.5 (S.32) of the CXTFIT manual (2.1) there are two coefficients for the two-region model. Thus, I have the question, what is the meaning of the calibrated value(e.g. Mu1 or Mu2) ?

Thanks a lot, Tobias.

model parameters:
INVERSE MODE NREDU
1 2 1
MODC ZL
1 5

***                BLOCK              B:        INVERSE          PROBLEM
                     ******************************************
MIT ILMT MASS
100 0 0
MNEQ MDEG
0 2
PHIM
0.5
***                BLOCK              C:        TRANSPORT        PARAMETERS
                     ****************************************
Mu1 Mu2
... 0.1 0.1
... 1 1



ntoride 03/09/2006 :  08:28:55
Tobias:
As shown in Table 3.1 in the manual, CXTFIT uses nondimensional parameters for
the nonequilibrium CDE. Table 3.5 summarizes possible simplifications for these
degradation coefficients in terms of three types of nonequilibrium models.

If we make these assumptions, dimensionless degradation coefficients mu_1 and
mu_2 are not independent. Hence CXTFIT internally relates mu_1 and mu_2 during
the iteration for parameter estimation depending on the MNEQ value in Block B (see
Table 6.4).

It is theoretically possible to estimate degradation coefficients. As far as I know,
however, I have never seen to succeed in properly estimating both nonequilibrium
transport parameters (beta, omega) and degradation coefficients at the same time. It
would be necessary to estimate the mobile-immobile parameters based on nonreactive
tracers.

Nobuo


Tobias, 04/13/2006 :  14:22:40
Dear Nobuo,
Thanks for your response!
But, there is still one thing I do not understand. In case that I have mobile-immobile
parameters (MNEG=0) from a conservative tracer and calibrate the decay coefficient
under the assumption that decay occurs only in the liquid phase (MDEG=2), the
calibration procedure yields to one dimensionless decay coefficient. But in table 3.5
are two dimensionless coefficients for the two region model (Mu1=Phi_m*Psi_l,
Mu2=Phi_im*Psi_l). Thus, I do not understand the meaning of the single calibrated
value (Mu_1 or M_2?) and how I can translate the calibrated value in a decay
coefficient with dimension [1/T].

Tobias.

Ntoride, 04/20/2006 : 01:11:36
Dear Tobias,
I am sorry for my late response.
>that decay occurs only in the liquid phase (MDEG=2), the
>calibration procedure yields to
>one dimensionless decay coefficient. But in table 3.5 are two
>dimensionless coefficients for the two region model (Mu1=Phi_m*Psi_l,
>Mu2=Phi_im*Psi_l).

When you choose this option, there is another additional input parameter PHIM (phi_m, theta_m/theta) as shown in Table 6.4 (p. 66). You can determine theta_m/theta based on the conservative tracer.

Because phi_im=theta_im/theta=1-theta_m/theta, phi_m and phi_im are known parameters. CXTFIT internally relates mu_1 and mu_2 assuming mu_2 = phi_im * phi_m/mu_1 during the estimation.

Nobuo


Tobias, 04/22/2006 : 23:07:27
Dear Nobuo,
Thanks for your response!
Thus, as far as I understand, I can estimate mu_1 [1/T] = (mu_1*v)/(L*phi_m), where mu_1 = decay coefficient under the assumption that decay occurs only in the liquid phases of the mobile and immobile region and is equal in both phases,
mu_1 [-] = calibrated dimensionless degradation coefficient,
v [L/T] = velocity
L [L] = length
phi_m [-] = theta_m/theta

Tobias.


## ID = 551, Boundary Nodes Error

Daisuke, 3/08/2006 : 15:12:59
I am receiving the error message, 'Error in FE mesh! Check that you do not have individual nodes on the boundaries.'
I am just trying to make a simple rectaglar mesh model with a small subsurface drainage pipe inside, however, I still keep getting the same error message and do not know how to solve this.
I made this shape with Meshgen and checked the geometry, and I don't think there is possiblity of overlapping nodes because it is a simple rectanglar shape.
I would appreciate any assistance on this matter.

Thank you,
Daisuke

Jirka, 03/08/2006 : 17:01:32
Daisuke,
You need to create the outer boundary with polylines and internal drain using a circle. Do not use any individual nodes to define, for example the corner of the domain. Line have their own nodes, and if you define additional individual nodes, then they are in the same location and overlap.

Jirka


Daisuke, 03/09/2006 : 13:29:37
Thank you!!
\\(^0^)


## ID = 552, Welcome

Jirka, 03/09/2006 : 19:23:36
Dear HP1 users,
Welcome to our new discussion forum that was set up to facilitate the technical support and mutual interactions of HP1 users. This discussion forum will operate similarly as our original HYDRUS-2D discussion forum.

If you want to contribute to this discussion forum you must first registered at this site. Registration is free and very simple. We recommend you to provide your full name and name of the company/institution for which you are working. Questions from fully registered members will be answered with higher priority than other questions.

Please, find below a brief description of HP1. More information can be found on www.sckcen.be/hp1 and in the manual of the code.
Good luck with the new software.

Jirka

HP1

A Coupled Numerical Code for Variably Saturated One-Dimensional Water Flow, Solute Transport and Biogeochemistry in Soil Systems.

The newly developed coupled reactive transport code HP1 (Jacques et al., 2005) is the result of coupling the variably-saturated water flow and solute transport model HYDRUS-1D (Version 2.0, Šimùnek et al., 1998) and the geochemical model PHREEQC (Version 2.4, Parkhurst and Appelo, 1999). HP1 combines the unique features of the two original models in one comprehensive tool by containing modules to simulate (1) one-dimensional transient water flow in variably-saturated media including a sink term accounting for root water uptake, (2) transport of multiple components with provisions for physical nonequilibrium (dual-porosity type) transport, (3) mixed equilibrium/kinetic biogeochemical reactions (aqueous

speciation, ion exchange, mineral equilibrium), and (4) heat transport. A full description of the processes, constitutive equations and numerical solution strategies are found in the manuals of HYDRUS-1D (Šimùnek et al., 1998), PHREEQC (Parkhurst and Appelo, 1999) and HP1 (Jacques and Šimùnek, 2005). The program numerically solves the Richards equation for variably-saturated water flow and advection-dispersion type equations for heat and solute transport. The flow equation incorporates a sink term to account for water uptake by plant roots. The heat transport equation considers transport due to conduction and convection with flowing water. The solute transport equations consider advective-dispersive transport in the liquid phase. The program can simulate a broad range of low-temperature biogeochemical reactions in water, soil and ground water systems including interactions with minerals, gases, exchangers, and sorption surfaces, based on thermodynamic equilibrium, kinetics, or mixed equilibrium-kinetic reactions.

Jacques, D., and J. Simunek, 2005. User manual of the multicomponent variably-saturated flow and transport model HP1: Description, Verification, and Examples (Version 1.0). SCK•CEN, Mol, Belgium, BLG-998, 83 p.

Parkhurst, D. L., and C. A. J. Appelo, 1999. User's guide to PHREEQC (Version 2) – A computer program for speciation, batch-reaction, one-dimensional transport and inverse geochemical calculations. Water-Resources Investigations, Report 99-4259, Denver, Co, USA, 312 pp.

Simunek, J., M. Sejna, and M. Th. van Genuchten, 1998. The HYDRUS-1D software package for simulating the one-dimensional movement of water, heat, and multiple solutes in variably-saturated media. Version 2.0, IGWMC-TPS-70, International Ground Water Modeling Center, Colorado School of Mines, Golden, CO, 202 p.

HP1 is available on request from www.sckcen.be/hp1. Jacques et al. (2003, 2005a), Jacques and Šimùnek (2005), and Šimùnek et al. (2006) demonstrated the versatility of HP1 on several examples such as a) the transport of heavy metals ($Zn^{2+}$, $Pb^{2+}$, and $Cd^{2+}$) subject to multiple cation exchange reactions, b) transport with mineral dissolution of amorphous $SiO_2$ and gibbsite ($Al(OH)_3$), c) heavy metal transport in a medium with a pH-dependent cation exchange complex, d) infiltration of a hyperalkaline solution in a clay sample (this example considers kinetic precipitation-dissolution of kaolinite, illite, quartz, calcite, dolomite, gypsum, hydrotalcite, and sepiolite), e) long-term transient flow and transport of major cations ($Na^+$, $K^+$, $Ca^{2+}$, and $Mg^{2+}$) and heavy metals ($Cd^{2+}$, $Zn^{2+}$, and $Pb^{2+}$) in a soil profile, f) cadmium leaching in acid sandy soils, g) radionuclide transport, h) long term uranium migration in agricultural field soils following mineral P-fertilization, and i) the fate and subsurface transport of explosives.

Jacques, D., and J. Simunek, 2005. Multicomponent - Variable Saturated Transport Model, Description, Manual, Verification and Examples, Waste and Disposal, SCK•CEN, BLG-998, Mol, Belgium.

Jacques, D., J. Simunek, D. Mallants, and M. Th. van Genuchten, 2003. The HYDRUS-PHREEQC multicomponent transport model for variably-saturated porous media: Code verification and application, MODFLOW and More 2003:

Understanding through Modeling, Conference Proceedings, ed. E. Poeter, Ch. Zheng, M. Hill, and J. Doherty, Int. Ground Water Modeling Center, Colorado School of Mines, 23-27.

Jacques, D., J. Simunek, D. Mallants, and M. Th. van Genuchten, 2005a. Long term uranium migration in agricultural field soils following mineral P-fertilization, Proc. Of "The 10th Int. Conf. on Environ. Remediation and Radioactive Waste Management, Sept. 4-8, 2005, Scottish Exhibition & Conference Center, Glasgow, Scotland, UK.

Jacques, D., J. Simunek, D. Mallants and M. Th. van Genuchten, 2005b. Modelling uranium leaching from agricultural soils to groundwater as yardstick in comparison with complementary safety indicators. Material Research Society 2005, Symposium on the Scientific Basis for Nuclear Waste Management, September 12-15, Ghent, Belgium.

Jacques, D., J. Simunek, D. Mallants, and M. Th. van Genuchten, 2006. HP1: A coupled numerical code for variably saturated water flow, solute transport, and biogeochemistry in soil systems, J. Contam. Hydrology, (in print).

Simunek, J., D. Jacques, M. Th. van Genuchten, and D. Mallants, Multicomponent geochemical transport modeling using the HYDRUS computer software packages, J. Am. Water Resour. Assoc., (submitted September, 2005, revised February 2006).

**ID = 553, Asking help on mass balance out vs input data**

Sherif Kamel, 03/09/2006 :  19:48:09
Dear users, I am using Hydrus 1D to estimate the infiltrated water to a soil layer in arid countries. I would like to check my model with professionals using this software. I used a surface water software to get the water depths into the stream to introduce it as water depths in hydrus as variable pressure head for the top BC and free drainage to the bottom BC. The soil profile is 10 m length and the soil is composed of 2 different types on the top 70 cm there is silt deposits and then the profile is composed of sand. The hydraulic properties of soil I entered according to the values of literature that I have in hand.
After I run the model, I have some questions concerning the mass balance output file, In the balance out file there are different values of area representing the length of the profile, the water volume is in units of length what does this mean in units of volume? also what is the difference of inflow and top flux? I observed that the inflow and the top flux is the same value for all print time interval but it is not the same value at the first time interval in other words at t=0? If I am interested in the infiltrated water to the soil at different times is inflow or top flux I have to choose? Also I would like to know,do i have to multiply the inflow and top flux (units Length/time)by the reach length and the reach width to get the infiltrated discharge in units of length cube/time (i.e: groundwater recharge rate)!!

I thank so much anyone who passed by these problems before and may help me to clarify things out.

Jirka, 03/09/2006 : 20:49:06
In one dimensional applications, the output balance is in units of lengh, i.e., cm of water. You need to multiply is by surface area and then you get volume of water.
Inflow in balance.out at time zero does not make sense. Inflow is calculated from the change of water in the profile in two successive times. At time zero this information is not available, and thus the code reports zero. Top flux is calculated from Darcy's law and thus can be slightly different than Inflow, but it should be very close.

Jirka


## ID = 554, Dual permeability options

Kgreaser, 03/10/2006 : 18:29:56
I have read the paper (with reference below) which states that several dual-permeability options are available in Hydrus1D but can't seem to activate them. The paper states the single-porosity non-equilibrium model of Ross and Smettem (2000) is available, as well as two dual-permeability models using Richards equation and the kinematic wave approach. In Hydrus, the last two options are listed but greyed out, ie I can't chose them. But the first option (Ross and Smettem) isn't even listed. And the manual only discusses the composite model of Durner and the modified van Genuchten option of Vogel and Cislerova, not any of the others listed above. I downloaded the new version 3.00, is there some other version with these options activated?
Thanks for your help.
Kelly

Paper ref: Simunek et al. 2003 Review and comparison of models for describing non-equilibrium and preferential flow and transport in the vadose zone, Jounal of Hydrology, v272, p 14-35.


Jirka 03/11/2006 : 21:36:20
I was working on this paper in 2002. At that time I developed new versions of both HYDRUS-1D and HYDRUS-2D that included these options to simulate nonequilibrium flow and transport. At that time I have been sharing these versions with many of my colleagues (basically, whoever asked for it; see selected references below), although they have never been officially released. In 2005, we have released version 3.0 of HYDRUS-1D and decided to focus on other new processes (e.g., colloid transport) and did not include these nonequilibrium features into this version at that time (partly because people were not successful in using these new features without our direct involvement). I plan to update version 3.0 with nonequilibrium features once I have a little time to do that (probably this summer; I have been too busy with HYDRUS (2D/3D) recently). To have more successful applications of these nonequilibrium features, we are preparing with Max and Sigrid Koehne the document that will accompany this new code with description of many examples and guidance how to (and how not to) use these new features.

Jirka

Zhang, P., J. Šimùnek, and R. S. Bowman, Nonideal transport of solute and colloidal tracers through reactive zeolite/iron pellets, Water Resour. Res., 40, doi:10.1029/2003WR002445, 2004.

Köhne, J. M., B. Mohanty, J. Šimùnek, and H. H. Gerke, Numerical evaluation of a second-order water transfer term for variably saturated dual-permeability models, Water Resour. Res., 40, doi:10.1029/2004WR00385, 2004.

Köhne, J. M., S. Köhne, B. P. Mohanty, and J. Šimùnek, Inverse mobile-immobile modeling of transport during transient flow: Effect of between-domain transfer and initial water content, Vadose Zone Journal, 3(4), 1309-1321, 2004.

Kodešová, R., J. Kozák, J. Šimùnek, and O. Vacek, Single and dual-permeability models of chlorotoluron transport in the soil profile, Plant, Soil and Environment, 51(7), 310-315, 2005.

Haws, N. W., P. S. C. Rao, J. Šimùnek, and I. C Poyer, Single-porosity and dual-porosity modeling of water flow and solute transport in subsurface-drained fields using effective field-scale parameters, J. of Hydrology , 313(3-4), 257-273, 2005.

Pot, V., J. Šimùnek, P. Benoit, Y. Coquet, A. Yra and M.-J. Martínez-Cordón, Impact of rainfall intensity on the transport of two herbicides in undisturbed grassed filter strip soil cores. J. of Contaminant Hydrology, 81, 63-88, 2005.

Köhne, J. M., B. P. Mohanty, and J. Šimùnek, Inverse dual-permeability modeling of preferential water flow in a soil column and implications on field-scale solute transport prediction, Vadose Zone Journal, 5, 59-76, 2006.

Köhne, S., B. Lennartz, J. M. Köhne, and J. Šimùnek, Bromide transport at a tile-drained field site: experiment, one- and two-dimensional equilibrium and non-equilibrium numerical modeling, J. Hydrology, (submitted September, 2004, in press).

Köhne, J. M., S. Köhne, and J. Šimùnek, Multi-process herbicide transport in structured soil columns: Experiment and model analysis, J. Contam. Hydrology, (submitted June 15, 2005, in press).

Gärdenäs, A., J. Šimunek, N. Jarvis, and M. Th. van Genuchten, Two-dimensional modelling of preferential water flow and pesticide transport from a tile-drained field, J. Hydrology, (submitted April, 2005, revised December 2005).

## ID = 555, Downloading Hydrus 3D

Ijsbrand, 03/10/2006 : 20:48:50
Hi Jirka,
I have tried to download Hydrus-3d but got a error while sending the data of the filled in Form.

Do you know why?
And what are the minimum requirements ro run the software (windows '98?)
regards,
IJsbrand de Haan


Mirek, 03/10/2006 : 21:41:41
Hi IJsbrand,
it seems to be working now, please try it once more.
There was an error caused by MS FrontPage and I've fixed it. May be you will have to press the "Refresh" button (or press "F5") to reload the fixed H3D download form.
Mirek


## ID = 556, Flux length unit

edsrlima 03/13/2006 : 16:48:20
Hi,
I´m modelling flow in a vertical plane into a collum and I would like to know why the flux lenght unit apear in cm² and not in cm?
Thanks,
Edson.


Jirka 03/13/2006 : 17:04:47
Flux across the boundary curve is equal to: Darcy flux * Length of the boundary = [L/T] * [L] = [L2/T]
Cumulative flux across the boundary curve is equal to: Darcy flux * Length of the boundary * Time = [L/T] * [L] * [T] = [L2]
Only Atmospheric boundary fluxes and root uptake are divided by boundary lenght, so that it results in [L/T].

Jirka


edsrlima 03/13/2006 : 17:53:08
ok,
Thanks.


## ID = 557, Numerical output of 1D

Elazar Bamberger, 03/15/2006 : 08:12:16
Hello
i am looking for the way how to get the numerical output of hydrus 1D. i want to work on the transient fenomena.
best regard
Elazar

Diederik, 03/15/2006 : 08:35:42
Elazar,
The output in HP1 generated by Hydrus is identical to that generated by the normal HYDRUS. It is located in the project folder and the files are named solute.out, obs_node.out, nod_inf.out etc (see Hydrus manual). These files contain information on the water content, temperature, pressure head, boundary fluxes, and (aqueous) concentrations of the master species (specified in the file species.in).

If you want other information (speciation, kinetics, composition of exchange sites, ...), you have to specify it in the phreeqc.in file using the keyword selected_output. For details, see the phreeqc manual.

Greetings,
diederik


**ID = 558, Inverse simulation - iteration paused**

Franta, 03/17/2006 : 20:17:54
During the inverse simulation my run for no apparent reason stops at the last time step (always the same time) and does not do anything even for a day or so. I think that I should receive a message that no additional reduction of sum of squares was achieved. Any suggestions?
Franta


Jirka, 03/18/2006 : 01:29:49
I do not know what can cause this. Is the program still running? You can see whether the file check.out is being updated (check the time when this file is created). Probably Marquard nonlinear optimization got into some infinite loop. Is one of the parameters approaching zero. Then specify some constraint such as 1e-5. Alternatively specify number of inverse iteration smaller than when it stops responding, and perhaps restart with the updated parameters.

I hope this will help you.
Jirka


**ID = 559, Boundary conditions**

Mary, 03/19/2006 : 04:35:49
I am trying to simulate solute transport for 31 mm columns. A suction pressure was applied at the bottom of the columns. So i have set the water flow bottom boundary condition as constant flux and the solute transport bottom boundary condition as concentration flux boundary condition. Does that make sense or a zero gradient for the solute transport bottom boundary condition would make more sense. I would really appreciate your help.

Jirka, 03/19/2006 : 19:51:40
If the bottom of the column is an inflow boundary, then concentration flux would be a correct BC. If it is an outflow boundary, then you can not specify solute flux BC, since you do not know the solute flux. You need to specify gradient BC.
Jirka


Mary, 03/19/2006 : 23:28:24
Actually, the bottom of the column is an outflow boundary. I tried it with gradient BC and the results make more sense.
Thank you very much fro your help.


## ID = 560, Output files

Silvia, 03/21/2006 : 10:04:46
Hello,
could you please help me again?
I insert in INPUT:
· vertical column, material type = sand, steady state, hydraulic head at top=1.2m, at bottom = 1m;
· colloid transport, attachment phenomena on one site (AttachSolid2=1E-5 sec-1), variables Fract. And Thlmob. equal to zero, upper boundary condition = constant flux, lower boundary condition = zero gradient.

Question 1
In NOD_INF.OUT there are the following columns:
Node, Depth, Head, Moisture, K, C, Flux, Sink, Kappa, v/KsTop, Temp, Conc(1...NS).
Then there are two columns with no name. What is their meaning?

Question 2
I can't keep the relative error in the solute mass balance (file BALANCE.OUT, variable CncBalR) low in the first half an hour of simulation. Is it normal/acceptable?
I tried, as suggested in a post, to use finer spatial and/or temporal discretization. For example I used:
Max Number of nodes = 200 nodes,
Initial Time Step = 6 sec
Minimum time step = 6 sec
Maximum time step = 100 sec
Absolute Concentration Tolerance = 1E-6 m/s
Relative Concentration Tolerance = 1E-6 m/s
Maximum Number of Iterations = 1000
CncBAlR goes from 90% to 26% in the first half an hour.

Question 3
In BALANCE.OUT "In-Flow" and "TopFlux" are very different ("In-Flow" is almost zero (E-10m/s)). Why?
Thank you very much,
Silvia

Jirka, 03/22/2006 : 17:42:46
Silvia,
a) Last two values in the nod_inf.out: Sorbed concentrations on first and second sorption (attachment) sites.
b) You should try to keep mass balance error below 2-3%. I can not say what causes mass balance error from available information. Concentration tolerances may be irrelevant if you do not have nonlinear sorption or blocking. Look also at the absolute values of the error. Since relative error is calculated based on solute fluxes across the boundary (see the manual) and not the mass in the system, the absolute errors may not be large.
c) While "Top Flux characterizes flux at the top boundary, "Inflow" characterizes the change of water in the system and thus includes both top and bottom boundaries (for steady-state, "Inflow" should be zero).


Jirka
Silvia, 03/23/2006 : 20:36:17
Hello Jirka,
thank you very much for your answers.

Question 1
I bother you again because I couldn't lower the relative error in the solute mass balance. I'm afraid I've misunderstood something important. Could you please have a look at the input data?
The input data are taken from "Modeling Colloid Attachment, Straining, and Exclusion in Saturated Porous Media"; Environ. Sci. Technol. 2003, 37, 2242-2250; Bradford, Bettahar, Simunek, van Genuchten, Yates.
Decline from Vertical Axes=1; Depth of soil profile=0.13m; Initial time step=0s ; Final time step=432000s ; Initial Time Step=6s ; Minimum time step=6s ; Maximum time step=100s ; Saturated water content (porosity)=0.378; Saturated hydraulic conductivity of sand-loam=1.22801E-005 m/s; Boundary conditions for flux=constant pressure head: head at top=1.2m, head at bottom=1.19m; Initial conditions=0 m in all nodes except for the first and the last ones; Absolute concentration tolerance=1E-7particles/m3; Relative concentration tolerance=1E-7; Maximum number of iterations=1000; Pulse duration=4620sec; Bulk density=1500 Kg/m3; Disp.=0.0024 m; Fract=0; Thlmob=0; DiffusW=10-8 m/s2; D-soil=0.00071 m; Ipsi2=0; Ipsi1=4; AttachSolid2=3.83333E-005 s-1; DetachSolid2=1.66667E-006 s-1; AttachSolid1=3.33333E-005 s-1; DetachSolid2=0; Upper boundary condition=concentration flux boundary condition (=3.86E+13particles/m3); Lower boundary condition=zero gradient.


Question 2
In solute.OUT the last column name is "cGWL" and its unit is [ML-3]. What is its meaning?

Thank you for your help,
Silvia

Jirka, 03/23/2006 : 20:51:26
Silvia,
I'm not going to reproduce this entire project. If you want me to look at it, then send it by email (both project folder and project.h1d file).

cGWL: If part of the soil profile is saturated, then the code calculates the average concentration over this part.


## ID = 561, Hydrus validation

Mary, 03/21/2006 : 21:17:14
I am simulating solute transport in 31 mm columns; where solution was added on top and drained from bottom by a suction pressure. The experiment was run for 12 days (each day adding 20 mL of solution), and each day the solution was collected from the bottom and analyzed for the contaminant concentration. I have set up HYDRUS to simulate 12 consecutive days independently, each running for 8 hours. I am having problems validating my model. I would like to get the concentration of solute drained from each column every day, so i can compare them with measure concentrations of the drained solution collected during the experiment. What would be the best way to get it? Shall i average the cbot (solute.out) at each hour for each simulation or should consider the cmean (balance.out) at the end of the simulation to be equal to concentration of the drained solution.
I would really appreciate the help.
Thank you.
Mary


Jirka, 03/22/2006 : 17:42:08
Mary,
You collect the outflow from the column and then determine the average concentration of the total outflow? If that is the case, then I would take the Sum(cvBot) value (cumulative solute flux at the bottom, in the Solutex.out) and divided it by the sum(vBot) (cumulative water flux at the bottom, in the T_Level.out file). The result should be the average concentration. cMean in the balance.out file is the average (arithmetic) concentration in the column (and thus the wrong value for your purpose).

Jirka


## ID = 562, Kv and Kh

Srilert, 03/22/2006 : 07:37:22
Dear all,
I am wondering for finding Kh and Kv by inverse solution in Hydrus 2d. Can it find these parameters and how?

regards,
Srilert


Jirka, 03/22/2006 :  18:05:38
HYDRUS has an option to fit an anisotropy ratio, but it is not supported by the graphical interface. Once you create the input, you need to manually modify the Fit.in file as follows (in bold):


iModel iHyst lAniz
0 0 t
THETAR THETAS ALPHA N KS l Aniz
0.008 0.35 .05 4.0 0.012 0.5 1.2
0 0 1 1 1 0 1
0 0.1 0 2.0 0.0001 0.001 0
0 0.6 0 7. 1.0 5.0 0


Jirka
Srilert, 03/23/2006 :  03:37:58
Dear Jirka,
Thank you for your answer. But i confuse a little bit, when I want to know this ratio that mean I assume this value first for example 1.2 and modified in fit.in file. And then change it when it's not fit. Right?

iModel iHyst lAniz
0 0 t
THETAR THETAS ALPHA N KS l Aniz
0.008 0.35 .05 4.0 0.012 0.5 1.2
0 0 1 1 1 0 1
0 0.1 0 2.0 0.0001 0.001 0
0 0.6 0 7. 1.0 5.0 0

and from above how much spaces between bold types and previous one.

Pls. explains me more. Thanks in advances.
Mr.Srilert Chotpantarat


Jirka, 03/23/2006 :  17:54:25
The value given below is the initial estimate (i.e., 1.2) of the Aniz parameter (ConAxx=ConAzz*Aniz). The number below it decides whether this parameter is optimized (1) or not (0, i.e., kept constant). Further below are constraints on optimization. The same as for the other parameters. It is free format, and thus it does not matter (empty spaces).
J.

**ID = 563, Water flow boundary conditions**

Mary, 03/22/2006 : 23:26:38
I have contacted regarding this experiment,but i need to be sure about the boudary conditions. I am trying to simulate solute transport for 31 mm columns. A 20 mL solution was dripped onto the soil surface from the top and a suction pressure was applied at the bottom of the columns to drain the columns. So i have set the bottom water flow boundary condition as constant flux and the top water flow boundary condition as constant head. The simulations seem to be fine, except for the moisture content. The experiments starts with moisture content of 0.394 then saturates till 0.5 (theta s) and goes back to 0.394 after all the solution drains after 17 hours. However, the results from simulation show that theta stays at 0.5 through out all the simulation; it does not go back to the initial conditions. Does the BC make sense, or is there something wrong with the initial conditions.

I would really appreciate the help.
Thank you.
Mary

Ftjin, 04/11/2006 : 20:15:38
Mary:
Have you tried implementing the Free drainage BC at the bottom? Let me know what result that gives you.
Jaspreert

**ID = 564, PHREEQC**

Ashmita, 03/23/2006 : 21:59:54
I am looking for a way to export data from PHREEQC to excel file. I have been trying to use selected_output and User_punch, though the file gets created, data from the phreeqc output files doesnt get exported. I have a 1-D transport model. I will appreciate any help on this issue
Thanks
Ashmita

Diederik, 03/24/2006 : 08:37:40
Ashmita
The file created by phreeqc using the keyword selected_output is not an Excell-fill but an ASCII-text file (Tab delimited). I usually open the file (e.g., output.sel) in Excell by using the Text Import Wizard (Delimeted , Tab of space). This file can then be saved as an Excell file.
Diederik
Ashmita, 03/28/2006 : 17:00:59

Thanks Diederik, I am able to create ASCII files and excel files from the output, however my problem is when I create the file, I can see headings etc but none of the numerical values (molality, si values etc) show up in the the file. I have a very simple 1D (15 cell) transport model, with chemical speciation. Is there a way I can post my input? Thanks again.
Ashmita


Ashmita, 03/28/2006 : 19:53:07
Hi Diederik,
I just figured out how to open the files. Thanks a lot for your help.
Ashmita


### ID = 565, Vapor transport and diffusivity coefficients

Alejandro, 03/28/2006 : 13:34:41
Well,so now I need to know about the water-vapour flux equation that is being used by Hydrus 1-D. Did you follow Philip and de Vries (1957)? Could you, please, send me this equation?

I thought this is the easiest way to separe at each node liquid from vapour component flux on water transport equation. New ideas are welcome!

Thanks in advance
Alejandro.


Jirka, 04/05/2006 : 00:33:40
Diff0 - diffusivity of water vapor in air [m2/s] (2.12e-5)
Adjusted for temperature as follows:
DiffT=Diff0*(TKelv/273.15)**2
and for tortuosity
Tau=ThetaA**(7./3.)/ThetaS**2
Diff=Tau*ThetaA*DiffT

The values can not be changed.
Jirka


Alejandro, 04/10/2006 : 20:29:06
I am using a Hydrus-1D version enhanced for vapour transport and
I would like to understand how the code is resolving molecular diffusivity. I am especially interested on the vapour diffusivity through superficial crusts. Which are the diffusivity coefficients being used by the Hydrus model for vapour in porous mediums? And, is it possible to change its values by empirical ones gotten on lab?.
Thank you

**ID = 566, Groundwater level and deep drainage**

julie j, 03/29/2006 : 15:13:16
Hello,
I have a problem building a model that includes an interface between the unsaturated and the saturated zones (from the surface down to the groundwater). Indeed, atmospheric data and observed water levels provide the required boundary conditions for the numerical model. As it is described in Exemple 2 of the tutorial, we must use the "deep drainage" condition.

Then we must specify the value of the parameters Aqh and Bqh, and the QWL0L, of the equation: q(h)=-Aqh.EXP.(Bqh.(h-GWL0L)) (as the exponential discharge of a watershed)

In the example 2, Aqh is negative, so q(h) must be positive…! And the GroundWaterLevel (GWL0L) is positive (+230cm); if this point is the bottom (lowest point) of the profile, and if the depth axis is positive upwards, shouldn't GWL0L be negative ?

I then wonder how to determine the value of these parameters: Aqh and Bqh. Indeed, this requires to calculate q(h), at the bottom for instance, and then to evaluate these parameters. However I tried to figure q(h) at the bottom, for the exemple 2, from the values of h at different time steps, (data from the file Nod_inf.out), but I did not succeed to find the same results for the flux.
So, then, if I am not able to calculate the q(h), how can I evaluate Ahq and Bqh? Or how can I figure q(h) at the bottom of the profile?

At last, I thought that we could use the lower boundary condition "Variable Pressure head"; but, a model with the upper atmospheric BC and the initial condition which fixes the free surface above the bottom of the profile, although the profile is initially saturated above the bottom, the soil becomes dry way too fast.

I hope my questions are clear and that my English is a correct "Shakespeare English".
Thank you very much for your help.
Julie


Jirka, 03/29/2006 : 22:58:51
Julie,
The correct function is without minus in front of Aqh:
q(h)=Aqh.EXP.(Bqh.(h-GWL0L))
You should be able to draw this function using T_Level.out file using vBot and hBot.
Jirka

You can find more on this function in:
Hopmans, J. W. 1988. Treatment of spatially variable groundwater levels in one-dimensional stochastic unsaturated water-flow modeling. Agricultural Water Management 15:19-36.

It is shown in this paper how the coefficients are determined.

In concept, they can only be determined from measurements of groundwater table and drainage discharge of the field. If you have these data, you have a very nicely defined lower boundary condition that provides fluxes that are calibrated from measurements. Likely, the magntitudes of these parameters characterize the drainability (magnitude as a function of time) of the field, and is likely field-specific, depending on your parameters (e.g. drain spacing, soil type, macropores presence, drain material, lateral flow of groundwater, leakage to deeper groundwater, etc.).

julie j, 04/06/2006 : 12:01:50
Jirka,
I have read the article, but I still have some questions..
I indeed can calculate the bottom flux, by the expenential law, with the hBot data from the output file.
However I want to calculate it from the Darcy law (as the bottom point belongs to the saturated zone, I use Ks and the pressure's gradient between the bottom point and another point which also belongs to the saturated zone). Using this law I can't find the vBot value. Why..??

Moreover what do you think about the "variable pressure head" lower BC instead of "deep drainage" lower BC?

Thank you very much...!!
Julie

Jirka, 04/06/2006 : 18:19:36
Julie,
If you know the bottom flux, then obviously you can specify it as a constant or time-variable flux BC. The problem in most applications is that the flux at the bottom of the soil profile is not known, and that is why we offer all these other BCs, such as free drainage, deep drainage, seepage face, etc.

Jirka

**ID = 567, Oil hydraulic properties using look-up table**

Besel, 03/30/2006 : 00:49:18
Dear Hydrus Team,
I am just wondering if there is a look-up table option for entering soil hydraulic parameters, since my measured properties are only poorly fitted by both the van Genuchten-Mualem and the Brooks-Corey model.
Benny

Jirka, 03/30/2006 : 01:12:46
No such option has never been made available.
Jirka

**ID = 568, Attachment, detachment and straining**

Raintree, 04/02/2006 : 04:46:58
Hello,
I'm very new to HYDURS1D. I want to use it to model colloids attachment-dettachment-straining process. However I was not able to find how to input parameters for straining function equation 3-28? Is there any sample input file for this problem?
Thanks,


Raintree 04/03/2006 : 04:32:48
Hello everyone,
I guess I pretty much figured out the previous questions and got some results. But I have one more question for the output:

I was able to find the colloids concentration in the liquid phase, which is in the file NODE_INF. But I was not able to find out the colloids concentration in the solid phase.

I would appreicate if anyone can give me some hints on it. Which output file? or do I need to set up anything to print out this variable?
Thanks


Jirka, 04/03/2006 : 16:58:26
It is in the same file, i.e., Nod_inf.out, just after the liquid concentrations.
J.

Raintree, 04/18/2006 : 19:14:10
Jirka,
Thanks! I found there are always two columns following liquid phase concentration in the NODE_INF. The first one is solid phase concentration. what is the second column after liquid phase concentration?
Thanks!


Jirka, 04/18/2006 : 19:27:20
HYDRUS allows you to have to kinetic sorption sites. Thus after the liquid phase concentrations, there are these two sorption sites. We usually use one for attachement/detachment and one for straining.
Jirka


Raintree, 04/18/2006 : 20:07:44
Thanks! Jirka. It makes sense now.
If my unit for the liquid phase concentration is (# particle/L), what will be the unit for solid phase concentration, is it #particle/gsand?
thanks!

Raintree, 04/18/2006 : 23:20:43
I have one more question. I was trying to use Attachment-Dettachment model. If I set not to optimize dettachment rate, it works fine, and it can give a reasonable result for attachement rate. If I choose to optimize dettachement rate, the program will stuck there after running for a while.
What is the possible mistake that I might make?
thanks!

Jirka 04/18/2006 : 23:20:48
Correct. J.

Jirka, 04/18/2006 : 23:28:03
It may be due to the Marquard-Levenberg inverse algorith. Sometime the optimized parameter, which is insensitive to available data, may tend to go to extremely large or extremely low values. When it goes to small values, the program tends to divide the value of parameters several times by 2, and then it never ends. It should help to set constraints on this optimized parameter.
J.

Raintree, 04/19/2006 : 02:51:51
Thanks Jirka! I set a lower boundary for the detachment rate, and it works well now!

Raintree, 09/14/2006 : 21:51:08
Hi,
I have a question about the inverse problem. Does the model only fit for the effluent concentration? Or it can fit for both effluent concentration and retention profile? If so, where should I put in the retention data?
thanks,
Raintree

Jirka, 09/14/2006 : 22:00:17
It can fit the retention data as well. The first column is the depth, the second is the sorbed concentration (S), type is 15, Position is equal to the Print Time.

Here is an example:

-0.611598 0.0920413 15 10 1
-1.92938 0.066118 15 10 1
-3.39709 0.120291 15 10 1
-4.85128 0.264327 15 10 1
-6.17309 0.336176 15 10 1
-7.47623 0.267779 15 10 1
-8.93477 0.237846 15 10 1
-10.4108 0.171135 15 10 1

-11.8792 0.11958 15 10 1

Jirka

## ID = 569, Negative concentrations

Jason, 04/03/2006 :  15:41:53
Hi - I'm simulating metals transport to the water table. For one of my metals, the concentration after about 300 years eventually reaches zero and becomes negative. Any thoughts why this would happen? Is the model just unable to accurately calculate solute transport beyond a certain time? This particular metal has a low Kd compared to the other 3 which don't become negative.

Thanks,
Jason

## ID = 570, Unsatchem mass units

Mvcallaghan, 04/04/2006 :  19:33:54
Jirka,
I have a few questions about the mass units used in the UNSATCHEM module of HYDRUS-1D. The mass and length units I have specified in my simulation are mmol and cm. In the UNSATCHEM manual the solute concentration units are listed as mmol/l and for adsorbed species as mmol/kg (in Table 12.8) elsewhere as molc/kg.

When I examine the mass balance in the CHEMBAL.OUT file, the tracer mass units are consistent with the units I specified for the total mass in mmol/cm^2. For the other solutes, the mass is 100 times larger than for the tracer. Is the mass for the other solutes in mixed units (in CHEMBAL.OUT) and what would they be?

Are the adsorbed species and CEC specified in mmol/kg? If so, what are the mass units in the CHEMBAL.OUT file? Is the bulk density specified in g/cm^3?

Mike

Jirka, 04/05/2006 :  01:16:19
In UNSATCHEM module of HYDRUS-1D, adsorbed and solid species and CEC are specified in meq/kg (or mmol of charge/kg), liquid species in meq/L. The bulk density should be specified in kg/L (which is the same as g/cm3)?
Then values in the CHEMBAL.OUT file (except for tracer, which is in m/cm2) (when the length units are in cm) are in meq/100 cm2.

Mvcallaghan, 04/05/2006 :  18:54:35
Many thanks for the quick reply, it was very helpful. My checks on the mass balance for solute concentrations check out fine with the addition of the factor of 100.

For adsorbed species, the mass units in CHEMBAL.OUT don't match up with my hand calculations. It is best illustrated with the following example.

1.4 meq/kg (adsorbed Na) * 1.62E-3 kg/cm^3 = 2.268E-3 meq/cm^3 of soil

For a 1000cm soil column: 2.268E-3 meq/cm^3 * 1000 cm = 2.268 meq/cm^2 = 2.268E2 meq/100cm^2

The total mass adsorbed that CHEMBAL.OUT contains is 2.27E5. There is a difference of a factor of 1000 in the results. What do you think the reason for the discrepancy is?

I also have a question about the equilibrium calculation that UNSATCHEM runs at the start of a simulation. I assume that the program performs an equilibration on the dissolved and sorbed species. I have been setting the solute concentrations for Na and Ca (binary system) using the Gapon equation (and CEC) to determine the equilibrium sorbed concentrations. The initial equilibrium procedure at Time=0 produces different results from my hand calculations. Generally the sorbed concentrations are close to what I have calculated, but the Ca in solution is orders of magnitude lower and the Na is moderately higher. Should the UNSATCHEM dissolved species and sorbed species concentrations always obey the Gapon equation and to what precision? The gapon coefficients are specified as K(Ca/Mg), K(Ca/Na), K(Ca/K) in the Hydrus interface. Are these coeff's given in terms of Ca selectivity over the other ion? For example the USSL Handbook 60 Gapon coeff for Na to Ca selectivity is 0.01475. Is that entered into Hydrus as an inverse e.g. 67.80?
Mike


Jirka, 04/06/2006 : 04:08:17
1.4 meq/kg (adsorbed Na) * 1.62 kg/L = 2.268 meq/L of soil
For a 1000cm soil column: 2.268 meq/L * 1000 cm * 100 cm2 (surface)= 2.268E5 meq/100cm^2
I know it is not the smartest way of doing it, but somehow I ended up with this.
How Gapon equations are included is described on page 69 of the manual (Eq. 6.17). Note that I'm using activities in the Gapon equation (not concentrations). Frankly, I do not know what these values should be, but your value seems increadibly large. I remember using values between 1 and 2 for K(Ca/Na).

Jirka


Mvcallaghan, 05/17/2006 : 19:13:19
It's been awhile since your response, but I wanted to add that your advice was very useful. Many thanks.

FYI for others: The use of activities in the "Gapon" type equation (which normally uses concentrations), has been referred to as being used to calculate the Vanselow coefficient. The paper "Sodium-Calcium Exchange with Anion Exclusion and Weathering Corrections" by Amrhein and Suarez, Soil Sci. Soc. Am. J. Vol.55, 1991, gives an excellent comparison between Gapon and Vanselow coefficients for various

soils. The Gapon coeff's (Na/Ca)for soils are on the order of 0.015 while the Vanselow coeff's are on the order of 1 to 3 (approx.).

Mike


## ID = 571, Vadose zone modelling

Ftjin, 04/04/2006
I am trying to model an unsaturated a clay sample, 1 meter in heigth.
The transient state model runs from time t=1sec to t=360000seconds
The initial condition is a set of suction potential varying with depth. At t>0, there is constant flux being applied at the top face.
The two side faces have zero flux boundar cobdition. The bottom face has zero suction potential boudary condition.
I am using the Van Genutchen curve parameters with an air entry value of -2cm.
For soem reason, the model continues to run and never proceeds beyond tiem step 1 second.
Any suggestions as to why this must be happening? and how to overcoe it?


Mvcallaghan, 04/05/2006
When you say "the model continues to run", what do you mean by that? Are you getting output to the screen in the DOS window? If you are getting output, is the model converging, or are you getting a convergence error message? Or is the model stopping at 1 sec. of runtime? More details would be helpful.


Ftjin, 04/11/2006
Jirka:
I am running a 1D colum simulation, with constant flux at the top face at all times, and 0 head at the bottom face. The soil is initially unsaturated. I am proving the initial condiiton in form of -ve suction potential. The model refuses to go beyond time step 1.
Any hints/Suggestions?


## ID = 572, Periodical pulse of solute

Alice, 4/05/2006
Dear Jirka
I am trying to simulate a periodical pulse concentration source inside a domain with HYDRUS-2D. Meaning that every 4 hours contaminants are pumped into the piezometer filter (the source). The transport inside the modeled domain takes place mainly by diffusion, both from the source into the medium and within the medium. I modeled the source as a second material, located somewhere inside the domain.

Could you, please, advice how to implement a cyclic contribution of solute?
Thanks a lot,
Alice

Jirka, 04/06/2006
Alice,
HYDRUS GUI does not support time variable internal sources (only time-variable boundary fluxes). However, in principal it would be possible to do that by manually altering the input files created by GUI.

a) In the boundary module create an internal source (using nodal recharge command)
b) Save as ascii
c) In the domain.dat file, the node will have Code=-1 and Q equal to specified recharge (positive for inflow). The node will not be listed in the Boundary.in file (since it is not on the boundary).
d) Close GUI.
e) Change Code=-3 and Q=0 in Domain.dat manually.
f) In the boundary.in file manually increase NumBP by one, add the node number at the end of "Node Number Array" and the width (=1) at the end of "Width Array" specify) and give the recharge value in the column rGWL (here it must be negative).

This should lead to pulsing of the internal source. Note that GUI may rewrite your boundary.in file and thus dissable the source again. You need to check that.
Jirka


**ID = 573, Topographic data**

Julie, 04/06/2006
Hi,
In order to create a new domain, is it possible to import topographic data in HYDRUS?
Thanks,
Julie


Mirek 04/06/2006
Hi Julie,
Yes, there are two ways how to do it:
1/ You can import one or more bitmaps (*.bmp files, for example scanned maps etc.) and use them as a "background" image:
a/ Insert 4 points that will be used as vertices of the rectangle defining bitmap position
b/ Go to menu: Insert -> Auxiliary_Objects -> Bitmap
c/ In the "Edit Bitmap" dialog pres "Load" and import a *.bmp file The bitmap should not be too large (<1MB) otherwise it could slow down the program during graphical input
d/ In the "Position" pane check "Visible" and then pick bitmap vertices
e/ Later you can hide/show this bitmap either in the "Edit Bitmap" dialog (Check Box "Visible") or in the Navigator->Tab_Show->Auxiliary objects->Bitmaps. Bitmap location can be changed easily by changing positions of definition points.

2/ If you have coordinates of points/lines created in a GIS you can import them in the following way:
a/ Go to Menu: Insert -> Domain Geometry -> Points -> Read_From_File

b/ The text file should contain coordinates in free format for example
for 2D projects:
1.1 3.400
1.4 3.566e-2
1.200e-1 100.
0.00012 134.
and for 3D projects:
1.1 3.400 1.0000
1.4 3.566e-2 1.5
1.200e-1 100. 2.0
0.00012 134. 2.5

We are going to create some video tutorials for this - it should be available on http://www.pc-progress.cz/Fr_Hydrus3D_Tutorials.htm within 2-3 weeks.

Regards
Mirek


Julie 04/06/2006
Thank you Mirek!
I will try to apply the second advice.
Julie


## ID = 574, High intensity rainfall events

Jvomacka, 04/06/2006
I am using Hydrus 1D to model an actual runoff event from a high intensity rainfall event, 10 cm/hr for 20 minutes. The initial conditions are very wet and the water table is at 40 cm. Ks for the soil is 5 cm/hr. My problem is that the solution is not converging. I have increased the max number of iterations to 8000 and am using a .2 node increment for a 200 cm column. I am using the Brooks and Cory solution with surface runoff which works fine for low insensity events. I have set the iteration table value to zero. Any suggestions?

Regards,
Jeffrey G. Vomacka


Ftjin, 04/11/2006
Jeffery:
Are you using the same units consistently? In your problem, u list cm/h and minutes.


Jirka, 04/17/2006
Jeffrey,
The problem with your project is that you have bottom boundary condition equal to no flow. Then you apply atmospheric BC at the surface with precipitation at a certain time. Your precipitation is larger than the pore space available to absorb this water,

and thus eventually you get fully saturated water, with additional precipitation that has nowhere to go, and the code crashes. You need to either use Free drainage at the bottom to allow water to leave the profile, or cut off precipitation before you reach saturation.

Jirka


Mvcallaghan, 11/29/2006
I'd like to reopen the discussion on this topic. Is it possible that the Hydrus code could be modified to apportion excess water at the atmospheric boundary to runoff when the soil profile becomes saturated? I have many cases where I am examining shallow water tables 0.5 to 1 m depth (fixed pressure condition on the lower boundary), and saturation of the soil column causes the model to crash. Increasing drainage on the lower boundary to within realistic drainage rates does not eliminate the occasional (e.g. wet spring) rainfall event that leads to saturation of the soil column. There must be a way of apporioning the excess rainfall to runoff and avoiding a model crash.
Jirka, what do you think?
Mike


## ID = 575, Discharge simulation

nasserga , 04/07/2006
Dear Jirka
I would like to represent the inflow boundary condition by a constant dicharge (L3/hr) insted of flux density(L/hr), How can I represent it. if the case is variable discharge, how can I represent it also?.
Thank you in advance and please accept my personal regards.

Dr. Gamal Khalil


Jirka, 04/08/2006
In two-dimensional simulations the flux across boundaries must always be specified as [L/T]. This flux is then in the code applied on the boundary of a certain length, and thus the total flux across the boundary is [L2/T]. You can view that as [L3/L/T], i.e. volume per unit length (in the third, or perpendicular, direction) per time.

Variable discharge needs to be specified as time variable flux, or atmospheric flux.
Jirka


## ID = 576, Do I need a 2D to do that?

Agua, 04/10/2006
hi,
i have rainfall and water elevation data of a bore.
i wanna model/extrapolate their behavior.

1. do i have to use another hydrological model to convert rainfall to runoff to input to hydrus? how i know the evaporation rate?
2. rainfall data i have is in mm yet the precipitation in hydrus is in L/T format, hence i need to specify whatever the cell cross section, does that must involve 2D model? if not, then how?
thanks
agua


Jirka, 04/10/2006
Aqua,
1. HYDRUS separates rainfall into infiltration and runoff based on solution of the Richards equation, i.e., taking into account soil hydraulic properties of the site.
2. I would assume that your rainfall data are in mm per some time interval, such as day.

Do you need 2D? I do not know. You did not provide any information that would be useful in deciding that.
Jirka


Agua, 04/11/2006
sorry i didn't make it clear.
i wanna simulate the groundwater recharge of a borehole(strata layers material known) by rainfall(also known in mm/d for 3 decades).
so later i am able to get the well yield aright.
one thing i get confused is the rainfall input in 1D is in L/T, then how i can know the volumn without knowing the cross-area, make sense?
anyway, could you give me some hint how to do that in 1D?
thanks a lot


agua, 04/11/2006
also more importantly, how can i estimate the time it takes for surface runoff(generated from rainfall data in any hydrological model)to get reach of the target aquifer (about 100m deep we have a drilling hole there to keep logging water elevation).
it is getting important now given queensland government legimatize the greywater replenish into groundwater but still concern about the quality issue...
thanks for help.


Jirka 04/11/2006
Aqua,
Flow to a borehole is at the minimum axisymetrical three-dimensional problem, and thus you need two-dimensional codes that allows axisymmetry. You can not do that with HYDRUS-1D, you need HYDRUS-2D.
Rainfall obviously falls at the soil surface. Rainfall is always measured in units of lenght per time, i.e., mm/d. Thus it rainfall is applied at the soil surface, the total flux

is "Rainfall" * "Surface Area", i.e., [L/T]*[L2, for axisymetrical problem], which is [L3/T].
The code will calculate how long it will take for water infiltrating into the soil profile to reach groundwater. It will not calculate what will happen with surface runoff, which is outside of the transport domain, and can not reach groundwater, but surface streams, ...
J.


agua 04/12/2006
thanks, jirka
i agree that flow into borehole needs 2D to simulate, yet i just wanna first run in 1D simply to get some idea about the time it takes, etc.
however when i firstly try rainfall recharge as a constant flux(rainfall should be time-variabe boundary but i just try the simple one) and set it in "upper boundary flux" but the model keeps showing "the numerical solution has not converged!", unless i try zero.
what is wrong?


agua 04/12/2006
later i tried to reduce the time step (min time step to 1e-9), it works but takes time, already half day has gone and i don't see the end so i shut it down.
basic info.: simulate 30m from ground, mostly sand, rainfall given but i still set up upper boundary as constant flux(0.5m/d), lower bc is free drainage. in graphic editor i set 200 nodes, -0.1 pressure head at both end.no other hassles.
can you see the problem?
thanks very much


Jirka 04/12/2006
Typical simulations with HYDRUS-1D do not take more than few seconds if the program is correctly set up. At present I'm doing 100 years simulations with daily values of precipitation/evapotranspiration fluxes, 30 m deep profiles of very contrasting soil layers, and these runs do not take more than few minutes.
Jirka


Agua, 04/13/2006
1.how do you input the rainfall data, is the precip. and evap. column directional(+/-)? do you use positive or negative to make sure it is seeping downwards?
a problem is when i input a constant flux is fine but when i start to input time-variable rainfall data, the solution cannot converge! why?

2. what is your initial pressure head in the soil profile? uniform or from top (maybe -1m) gradually to bottom(0 m at water table in my case, since recharge stops at the table, right?)
thanks a lot

**ID = 577, Drip simulation once more**

Nesr, 04/10/2006
Dear Proffessor
I am a PhD student working in both Surface and Subsurface drip irrigation PhD topic.

I would like to ask about simulating Drip Emitter Point Source,
However, I read some answers in the forum about conversion from FLOW to FLUX but dividing the discharge (L3/T) by area (L2) is a problem in drip irrigation as The Dripper's wetted area is variable with time (increasing with time) so the flux will decrease with time according to the soil type, How can I represent a 4L/h emitter for example???

Another Problem is that I don't know how to apply values of variable flux BC!
Thanks in advance for your reply

Jirka, 04/14/2006
In HYDRUS-2D you need to specify the surface area and then the flux across this surface areas. This surface area must be constant during the simulation.

That is clearly not very suitable for drip irrigation. Thus for one of my project I have modified HYDRUS to allow for a time-variable ponded boundary condition by computing wetted surface area. This was done by switching from a Neumann (flux) to a Dirichlet (head) boundary condition if the surface pressure head required to accommodate the specified emitter flux for a surface node, is larger than zero. A sufficient number of surface nodes are switched in an iterative way until the entire irrigation flux Qo is accounted for, and the radius of the wetted area is obtained. Since the infiltration flux into the dry soil is larger for early times, the wetted area continuously increases as irrigation proceeds.

However, this option is not yet available in the regular HYDRUS-2D.

**ID = 578, Boundary condition in Hydrus 1D**

Ftjin, 04/11/2006
The HYDRUS-1D help menu lists the folowing upper boundary conditions:
Constant pressure head
Constant flux
Atmospheric boundary condition with surface layer
Atmospheric boundary condition with surface run off
Variable Pressure Head
Variable Pressure Head/Flux

Out of the which only the Atmospheric and Variable pressure head/Flux conditions are highlighted and explained in the menu. Does this imply that one cannot use a constant flux BC in Hydrus1D?

Jirka 04/12/2006
In the help file we explained only selected boundary conditions, since we thought that some are so clear that there is nothing to explain, nothing to add. What can you add to the constant pressure head BC - it is constant in time, and it is a pressure head BC? Obviously all boundary conditions listed can be used in HYDRUS.
J.

## ID = 579, Brooks and Corey model with surface runoff

Jvomacka, 04/11/2006
Has anyone run Hydrus 1D with the Brooks and Corey model with the upper boundary condition set at "atmospheric BC with Surface Runoff"? I can get Hydrus to run with the upper boundary condition set on "atmospheric BC with surface layer" but not with surface runoff. When I switch from "surface" layer to "surface runoff", the model will not converge. I have 200 cm column with 1001 nodes. Any suggestions?

Regards,
Jeffrey G. Vomacka

ftjin 04/11/2006
What do you mean by the model does not converge? Does it show the message' model does not converge' or it cotinues in the iterative mode without actually moving forward in the time step?
Jaspreet

jvomacka 04/11/2006
Jaspreet,
The model runs until there is a rain event which should cause runoff, but instead, the message shows solution does not converge. I let it run for a minute or two, but the model never converges.
Regards,
Jeffrey G. Vomacka

quote:
What do you mean by the model does not converge? Does it show the message' model does not converge' or it cotinues in the iterative mode without actually moving forward in the time step?
Jaspreet

ftjin 04/11/2006
Have tried smaller time steps? What time step are you using?

Ftjin, 04/13/2006
What sign convention are you using for the rainfall (flux).Remeber that in ase of HYDRUS1 D, the vertical axis is poitive upwards, so you rainfall should have a negative sign. Let me know if this works.
Thanks
Jaspreet


Jirka, 04/14/2006
This is really not true about Precipitation, evaporation, and transpiration fluxes. They should be all entered as positive values.
J.


## ID = 580, Root distribution parameters window

Gerard Arbat, 04/12/2006
Hi
Somebody knows how to open the dialog window for root distribution parameters (Vrugt et al, 2001, 2002), as it appears in figure 30, page 63, in the User Manual?
Thanks,
Gerard


Mirek 04/12/2006
Hi Gerard:
In the main menu go to: Edit -> Domain Properties -> Parameters for Root Distribution...
This command is enabled only if the "Root Water Uptake" check-box is checked in the "Main Processes" dialog.
We will add this command also to the Navigator and Edit Bar in the next version so that it was more easy to find it.
Regards Mirek


Gerard Arbat,  04/12/2006
Thanks Mirek,
It works.


## ID = 581, Mass balance error, negative concentrations

Birdie, 04/12/2006
I am getting negative concentrations when running a simulation for long time intervals (200 - 2000 years). The mass balance errors indicated that for some of the contaminants being simulated the error will rise above 5% at about 70 years. Are these errors the reason that the negative concentrations are being predicted? Is this to be expected and are extra long time simulations not useful?

Jirka, 04/12/2006
Read the section about numerical oscillations (and stability) in the manual and see whether using suggestions given there will help you.

Jirka

## ID = 582, hCritA value

Agua, 04/13/2006
when setting up the time-varibale boundary condition(rainfall data input), what does hCritA actually mean? and how the hCritA value changes the result?
i cannot understand what the manual says.
thank you.

Jirka, 04/13/2006
It is the minimum pressure head allowed at the table. When this value is reached than evaporation is reduced from potential to actual values. I usually use values of about 150 m. It can be precisely calculated from the air humidity.
J.S.

## ID = 583, Observaion nodes

Srilert, 04/18/2006
Dear all,
when i simulation and put observation nodes in domain, I found that simulation can be run already but I cannot to open output of observation nodes. And then, I went to open ObsNod.out, I found below title "hnew" it show "*******..." like this. That I think it is causing that cannot show graph of output at observation nodes. How can I solve this problems of simulations.
Thanks in advances.

Jirka, 04/18/2006
Your simulation was clearly unsuccessful. This appears (*****) when the number is larger than the allowed format, which for pressure heads is f13.3. Thus your pressure head numbers are unrealistic.

Jirka

## ID = 584, Source amount

Jason, 04/18/2006
Hi all,
When modeling solute transport, is the source considered finite, or is there an unlimited supply that keeps being introduced throughout the course of the run. After

running the model for an extended period of time, there is a peak output concentration which declines after a certain point. This to me seems like the source is finite. Thanks.

Jason


Jirka, 04/18/2006
Jason,
There is a variable "Pulse Duration" in the "Solute Transport - General Information" dialog window. For times larger than "Pulse Duration" or time-invariable solute BCs are turn to zero.

Jirka


### ID = 585, Meshgen & Hydrus

KlausR, 04/18/2006
Hi all,
I am using Hydrus 2D v 2.102 with meshgen. I have constructed a domain with approximately 150 rectangular shaped internal curves that are randomly distributed throughout the entire domain.

Meshgen fails with the message that the domain is ill conditioned and suggests adding internal points.

Adding points to the internal curves and boundary domain does not help. Infact I have observed cases where adding points causes meshgen to fail for situations where meshgen worked prior to refinement.

Meshgen will work when I reduce the number of internal curves to about 50 positioned near the center of the domain. But I wish to have these internal curves throghout the domain.

Thanks in advance for any suggestions,
Klaus


Mirek, 04/18/2006
Hi Klaus,
please send me the project. I'll look at your domain and then we can discuss other details.
Regards
Mirek


Mirek, 04/19/2006
Hello Klaus,
Your domain is really interesting and seems to be very good for testing of Hydrus/Meshgen performance. At first glance I did not see any trivial error there. FE-

mesh generation was very slow and fundamental triangulation really failed. I tried to open this project in our new HYDRUS 2D/3D version BETA.02 but there was an error in import. Therefore I opened it in our latest version BETA.03, that is to be released this week, and then I was able to find several problems:

1/ At the bottom-left corner of your domain (x,y = (0,0)) you have 3 overlapping points. You will see them if you delete boundary curves (the left and bottom curve). It would be good to select these points with rectangle and delete them. But these points are not the reason why mesh generation fails (MESHGEN ignores them).

2/ The real problem is that some of your internal rectangles are very close – they are almost touching each other. Meshgen uses double precision for calculation of fundamental triangulation but according to my experience it is not so difficult to create a domain where the triangulation fails (I don't want to go into details). We call such domains "ill-conditioned". In your case you will need to delete or move some of those rectangles mentioned above so that they were not so close (min distance should be about 100 cm).

3/ I think that even if you generate successfully the FE-mesh then it will not be easy to finish your analysis – your mesh will be too large. I was able to generate FE-mesh for your domain in HYDRUS 2D/3D and I got over 200 000 elements. I think that HYDRUS calculation on such large FE-mesh will be very slow – I'd recommend you to contact Jirka Simunek and ask him about his opinion.

4/ The last remark: I would recommend you to upgrade to the new HYDRUS 2D/3D. There are several reasons for that (apart from many other new features in new HYDRUS):

a/ New HYDRUS will give you more tools how to find problems/data inconsistency in definition of your computational domain. For example now we are working on option that would display the position where the triangulation failed. This would help you to find exactly those internal rectangles responsible for mesh generation error. I'm not sure if we manage to finish this option in BETA.03 but it definitely will be in next BETA.04.

b/ If you delete an internal rectangle in old Hydrus-2D then you loose your previous definition of FE-mesh points on boundaries. This is very frustrating because in your case this definition is very time-consuming and you have to repeat it several times (because you don't know what internal rectangle you need to delete). New HYDRUS doesn't delete your previous definition of FE-mesh refinements if you change your domain and it is also much easier to define required size of finite elements on your internal curves. You can just set the targeted FE-size to S=150 cm and that is all, this size is used automatically on all internal curves.

c/ We are going to optimize FE-mesh generation in new HYDRUS so that it run smoothly and fast even for such complex domains as you have. Unfortunately we are not able to make this optimization in old Meshgen.

I'm going to test your domain in new HYDRUS 2D/3D and if I find something that could help you to solve your problems in your current Hydrus-2D/Meshgen I'll let you know.

Regards
Mirek


KlausR, 04/19/2006
Mirek,
Thank you for the very fast and very thorough response. I will try your suggestions and let you know how it goes:
1) remove overlapping points at 0,0
2) move closely spaced internal boundaries to a min of 100 cm separation
3) reduce number of nodes on the internal boundaries to reduce the total number of elements.

I have tried the 2d/3D beta 02 version and am very impressed by improvements and features of the code. I have stayed with the 2D version for the time being b/c I am familiar with this version, but will gradually migrate to the new version.

One minor quibble with the new 2d/3d code is the limitation of 10 isolines in the output display. I could not see a way to increase this, and 10 isolines does not provide enough resolution for my domain which has multiple material properties with different saturation ranges. Any way to increase this?

I too tried unsuccessfully to import the domain into beta 2 version. I am looking forward to trying it on the beta 3 version.

Klaus


Mirek, 04/19/2006
Klaus,
there are another two overlapping nodes on internal boundaries with coordinates:
x = 34418.60
y = 3529.04

x = 46698.50
y = 5144.05
I did not notice this in new HYDRUS because after receiving the warning about overlapping nodes at x,y=(0,0) I used function "Repair domain" that fixed all overlapping nodes automatically and then mesh was generated correctly. Later I was interested why old Meshgen fails even if I removed duplicated nodes at 0,0 and now I think that the main reason is the two overlapping nodes mentioned above.

Isolines:
I think that we could draw some "intermediate" isolines between each pair of main isolines (main isolines = current state). Standard would be N=1 (current state) but user

could specify how many isolines he wants - for example N=5 or 10,... I'll discuss it here and may be we will add it into our "to do" list.

Regards
Mirek


Mirek, 04/20/2006
Klaus,
We just released Beta.03. You can download it and use it for detection of overlapping points in your project. You can do it in the demo version: import/convert your old Hydrus-2D project and go to menu -> Tools -> Check geometry.

We are going to use your complex domain as a testing example for optimization of new HYDRUS and I believe that we will be able to make HYDRUS significantly faster than it is now...

Regards
Mirek


**ID = 586, Drip irrigation flux!!**

Nesr, 04/19/2006
Dear Prof. Simunek
concerning the drip source representation in hydrus2D

I tried to solve such problem by another way and I wonder if it is correct. Simply I defined an axi-symetric Vertical boundaries of 50x75 cm, I defined all boundary nodes as "No-Flux" condition except the upper left 2 nodes, which I defined a "Constant-Flux" BC of 1 cm/min, I ran the simulation for 60 min. then I read the "Mass Balance Information" module, where I found the term "InFlow [V/T]" in cm3/min.

I re-ran the program several times with different fluxes (for the same two upper left nodes), and I found an equation for computing the Flux to be input f(cm/min), for a known discharge value q(Liter/hr) that is f= 0.0035+1.074 q.

Am I right ??
Is that "InFlow [V/T]" represents all the volume of water in the grid or half of it? or else?

You, kindly, offered to send me some of the drip irrigation examples and some codes, I am waiting for it.

Many thanks for you
El-Nesr

Jirka, 04/19/2006
I have sent you the special version of HYDRUS with special dynamic surface boundary for drip irrigation to the email-address drnesr@gmail.com on 4-15, and resent it today.

If all the other boundaries are not flow boundaries, then indeed Inflow is the infiltration flux from the drip. However, mOre precise values are in v_mean.out and CumQ.out. It is in cm3/time for the entire domain for axisymmetrical applications, and in cm3/cm/time for two-dimensional applications.

The infiltration flux should be exactly equal to specified flux multiplied by surface area. What surface area is associated with your two surface nodes can be found in Boundary.in.

Jirka


Jirka, 04/19/2006
Emails that I send you are comming back with a message:
The following addresses had permanent delivery errors ----- drnesr@gmail.com


Jirka, 04/19/2006
I guess Google email does not allow files with certain extension. Thus I rename "zip" with "piz" and "exe" with "ex". When you get the email you need to rename it back.
Jirka


Nesr, 04/20/2006
You are right sir, Google mail does not accept any executable files as attachment neither zip and zip related attachments except the .rar format till now
The only way to do so is to rename the extensions to an unknown file type.
The last email you kindly sent have arrived successfully
I received the attachment and renamed the piz to Zip and the ex to exe. it works well but you said in your mail that [["Description of this example is in the attached paper"]] but I receive no papers in the attachments.
Please send me the paper or the description of the problem.

best regards
M. B. El-Nesr


## ID = 587, CXTFIT one-site model question

Rosie 04/19/2006
I am trying to run an inverse problem using the one-site nonequilibrium CDE model, and I keep getting this error message (flashes on the screen very quickly) prior to the output display:
run-time error M6103: Math
- floating point error: divide by zero

Then when I look at the output graph, it shows only my data, no fit curve. I have run the same exact data through the two-site and two-region models, and I can't figure out what's wrong. It seems like I have messed around with every possible variable in the simulation.
Any ideas?


Rosie, 04/19/2006
I should mention I had no problems getting fit curves to appear for the two-site and two-region models.


Ntoride, 04/20/2006
Please send me your input file (SANMOD: the project folder with the cxt file, CXTFIT: cxtfit.in). I will have a look at the input parameters.
Nobuo


## ID = 588, pH effect

Julie, 04/22/2006
Diererik,
Just for curiosity, I would like to know how to implement the pH effect in HP1.
If I want to know this effect on Cationic Exchange Capacity, I suppose that I have to use the log K of the equation $H+ + X- = HX$ (right?) but if I want to adapt an eventual Anionic Exchange Capacity as a function of pH, how could I do that?
Many thanks,
Julie


Diederik, 04/25/2006
Julie,
I have never done that, but I suppose that you can define a anion complex (Y-) and then use the reactions $Y- + OH- = YOH$ with corresponding K-value to describe the pH dependence.
greetings,
diederik


Julie
Senior Member, 04/25/2006
Hi,
Thank you.
Maybe I will have the opportunity to do that in the future.
Julie

**ID = 589, Problem in solute tranport simulation**

Srilert, 04/25/2006
dear all,
when i run Hydrus 2d, in starting run it's O.K. but when pass for some minutes, window show the time is the same value llike
Time Itw Itcum..........
90.1424
90.1456
90.1456
90.1456
90.1456
90.1456
90.1456
and so long time and not change.
How should I do to solve this problem?
Thanks in advances.
Lertc


**ID = 590, Simultaneous calibration of different parts ofa m**

Kalaerts, 04/25/2006
Dear all,
I 'm a student writing my thesis about the evaluation of water captation techniques and therefore I am trying to inversely calibrate a model of an experimental set up using Hydrus-2D. The experiment consists of a rainfall simulation on an experimental plot, with an infiltration furrow at its lower end. The model consists of three parts: a simulation of the 20-minute rainfall simulation with the filling up of the furrow, a second part simulating the falling of the water level after the simulation and the evaporation on the field and a third part only simulating the evaporation on the entire field and furrow during the 5 days after the rainfall simulation. For the entire period, TDR-measurements of the moisture content at different depths and different distances of the furrow are available, as well as pF-curves and the estimated van Genuchten parameters at the same depths. I am now trying to calibrate the model using the soil moisture content measurements and assuming the van Genuchten parameters to be correct (to avoid too many parameters to be estimated), but even when using a batch file as explained in 4.13 in the manual I don't manage to obtain results for the simulation as a whole, I can only get estimated Ks and n values for the three simulations separately. Is there any command that can be used in the batch file to let the three parts run sequentially during the calibration? Or will I have to calibrate the model by hand?
Thanks,
Katrijn

Jirka, 04/30/2006
Katrijn,

I have not answered this since it was not exactly clear to me how you defined your problem. If I understand correctly, you try to simulate three time intervals with different boundary conditions:
a) Atmospheric BC (rainfall) on the soil surface and time-dependent (increasing) pressure head (that you specify) BC in the furrow.
b) Atmospheric BC (evaporation) on the soil surface and time-dependent (decreasing) pressure head (that you specify) BC in the furrow.
c) Atmospheric BC (evaporation) from both the soil surface and the furrow.

HYDRUS-2D could not solve problems where the type (not only the value) of the boundary condition changed with time. In HYDRUS-2D, you could only change the value (i.e., flux or head), but the time had to remain the same during the simulation. To change the type of BC, you had to split the simulation into several parts.

We have tried to overcome this problem in our latest version of HYDRUS (i.e., HYDRUS (2/3D)) that we released about two month ago. I believe that with this new version you would be able to solve your problem. (if you send me your three project, I can have a quick look at it). See below the description of new system-dependent boundary conditions that I copied from the technical manual of HYDRUS (2D/3D).

Jirka

In addition to system-dependent boundary conditions available in version 2.x of HYDRUS-2D, several new options are available in HYDRUS:

a) While in version 2.0, all boundary conditions (i.e., fluxes or pressure heads) changed in abrupt steps, the new version allows boundary pressure heads to change smoothly with time. Abrupt changes in the pressure heads lead to sudden changes in fluxes, while smoothly changing pressure heads provide smoothly changing fluxes. An example of such a boundary condition is the water level in a stream or furrow.

b) While version 2.0 only allowed either time-variable pressure heads or time-variable fluxes on a particular part of the boundary, the new version allows boundary conditions to change from variable pressure heads to a zero flux and vice-versa. This boundary condition can be used for example for a disc permeameter where the specified head changes to a zero flux during time periods when the permeameter is re-supplied with water. The zero flux is initiated by specifying a value larger than 999999.

c) When a time-variable pressure head boundary condition is specified along a boundary, then the specified value is assigned to the lowest nodal point of a particular boundary, while pressure heads at other nodes are adjusted based on the z-coordinate. When this option is selected, then nodes with calculated negative pressure heads are not associated with a Dirichlet boundary condition, but rather with a zero flux. A fluctuating water level in a stream or furrow is an example of this type of boundary condition. While positive pressure head values are below the water table, negative values occur above the water table.

d) This is similar to c) except that an atmospheric boundary condition is assigned to nodes with negative calculated pressure heads.

e) This is similar to c) except that a seepage face boundary condition is assigned to nodes with negative calculated pressure heads.

f) When this type of system-dependent boundary condition is selected, then HYDRUS treats the time-variable flux boundary conditions similarly as atmospheric fluxes. This means that pressure heads have two limiting values, with the maximum pressure head equal to hCritS and the minimum pressure head equal to hCritA.

g) While in version 2.x of the code, the flux across the nonactive part of the seepage face was always equal to zero, the new version can apply atmospheric boundary conditions on a nonactive seepage face.

h) When heat transport is simulated simultaneously with water flow and atmospheric boundary conditions, then snow accumulation on top of the soil surface can be simulated. The code then assumes that when the air temperature is below -2 C all precipitation is in the form of snow. When the air temperature is above +2 C all precipitation is in the form of liquid, while a linear transition is used between the two limiting temperatures (-2,2). The code further assumes that when the air temperature is above zero, the existing snow layer (if it exists) melts proportionally to the air temperature.

Kalaerts, 04/30/2006
Dear Jirka,
many thanks for your help, the problem is exactly as you describe it and I indeed think that this new version will be able to solve my problem. I will send my files to you, so that if by any chance you have some time you can look at it.

Thank you very much in advance,
Katrijn

Jirka, 05/03/2006
Katrijn
I have converted your three HYDRUS-2D projects into one project in HYDRUS (2D/3D) and I could indeed, after small modifications, run its all three stages during one single run. I'm sending you an email with some inverse results.
Jirka

**ID = 591, Re-install with new version**

Nursetiawan, 04/26/2006
Dear Mirek
If I want to change the previous version of Hydrus 2D/3D with the newest one, should I delete the older version first from my PC/Notebook? And how about activation code?
Thank you.
Best regards,
Nur - UK


Mirek 04/26/2006
Hello Nur,
the best way is to install new version on the same hard disk but to a different directory (for example HYDRUS_03). Then your authorization should remain working. Later you can delete the directory with old HYDRUS.
If you need to install HYDRUS on a different hard disk or computer then you will need new activation codes. In that case you should deactivate old HYDRUS (menu Help -> Hydrus License and Activation -> button "Deactivate") and send us the deactivation file "DeactH3D.lf" with your new request codes.

Regards
Mirek


Nursetiawan, 04/26/2006
Thank you Mirek for the quick respond.
I will try it.
Regards
Nur


**ID = 592, H3D/release wells at an embankment**

Hector, 04/26/2006
Hello out there,
I have been performing some inspections with FEFLOW-3D on release wells at the inclined surface of an embankment which is an explicit 3D-problem and would like to check against the performance of HYDRUS 3D professional. So if you need an beta-tester....

Regards
Héctor


Jirka, 04/27/2006
Nice try, Hector. We already have quite a few Beta testers. Thanks for the offered help.
Jirka

**ID = 593, Question: inverse analysis of a one-step outflow E**

Lifang, 04/26/2006
For my drainage experimen, the upper BC open to at atmasphere, lower BC at the tesion of -160cm.Inition condition is saturated. I want to get the hydraulic conductivity VS head with my collected outflow data

how to select Upper Boundary Condition, atmosphere BC with surface layer or with constant flow rate 0.
Plus, I did not find where to specify the lower BC -160cm.
Thanks a lot to whom will help me

Jirka 04/27/2006
Examples for the onestep and multistep experiment are installed together with the HYDRUS software. Check them out. Description of these examples is given in the manual of HYDRUS-1D and in:
Hopmans, J. W., J. Šimùnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Do your homework before asking questions in this discussion forum!!
Jirka

**ID = 594, Variable water content in solute transport**

Srilert, 04/29/2006
Dear all,
I try to simulation of contaminant(heavy metals) from soil surface leak into groundwater by varying initial water content at each node from 25%, 50%, 75% and 100% of water content with flux about 0.08mm/day . And then, I observe at each depth, I found that at each depth fror example, 1 m from surface, it has the same concentration at the same time , in other word it does not have affect of water content. I think, it should be different by common sense. Could u pls. help me ?
Looking forward for your help.
Lert

Jirka, 04/29/2006
Lert,
I think that results of simulations are correct. Although you will have a completely different time of arrival of the water content front (earlier arrival for wetter profiles), concentration front should arrive at the same time. You can calculate arrival of the concentration front to depth L as: t = Depth / Pore Water Velocity = Depth * Water Content / Darcy's flux. Since Darcy's flux is the same for all applications, so will be the final water content (irrespective of the initial water content), and thus t is the same for all simulations.
Jirka

**ID = 595, Calibration**

Kalaerts, 04/29/2006
Dear all,
I 'm so sorry for bothering you again with the same question, but I really would appreciate any kind of answer that could possibly help me, because right now I'm doing the calibration of the three-part model by hand and it's costing me an awful lot of time without any nice results upto this moment. Since I have to hand in my thesis the 22nd of May anything that could help me in saving some calibration time would be very useful. I hope my problem was explained clearly, thanks a lot for helping me out!
Katrijn


Jirka, 04/30/2006
Katrijn,
I have not answered this since it was not exactly clear to me how you defined your problem. If I understand correctly, you try to simulate three time intervals with different boundary conditions:
a) Atmospheric BC (rainfall) on the soil surface and time-dependent (increasing) pressure head (that you specify) BC in the furrow.
b) Atmospheric BC (evaporation) on the soil surface and time-dependent (decreasing) pressure head (that you specify) BC in the furrow.
c) Atmospheric BC (evaporation) from both the soil surface and the furrow.

HYDRUS-2D could not solve problems where the type (not only the value) of the boundary condition changed with time. In HYDRUS-2D, you could only change the value (i.e., flux or head), but the time had to remain the same during the simulation. To change the type of BC, you had to split the simulation into several parts.

We have tried to overcome this problem in our latest version of HYDRUS (i.e., HYDRUS (2/3D)) that we released about two month ago. I believe that with this new version you would be able to solve your problem. (if you send me your three project, I can have a quick look at it). See the description of new system-dependent boundary conditions that I copied from the technical manual of HYDRUS (2D/3D) in your previous question in this discussion forum.

Jirka


**ID = 596, Interface water flow**

Nasserga, 04/29/2006
My Dear Jirka
Please, I need some information to simulate the water flow through the interface between two soil layer. is HYDRUS-2D can simulate this problem?if yes, how can I do it?
Thank you in advance and please accept my personal regards.
Dr. Gamal Khalil

Jirka, 04/30/2006
Dear Dr. Khalil,
HYDRUS-2D can certainly simulate flow and transport in a soil profile with multiple soil layers (see Test7 from workspace Direct installed together with HYDRUS and described in the manual). If you want to find out what is the flux between two domains, then this is not reported in the output. It can, however, be easily calculated from various information in the output files. In the balance.out, you can find water balance for the entire transport domain and specified subregions. In the v_mean.out and CumQ.out, you can find actual and cumulative fluxes across boundaries of different types. By combining this information, you can calculate the flux between the two regions. The procedure is described in detail in the book of David Rassam.

Jirka

## ID = 597, Postdoctoral fellow in watershed hydrology

Alifares, 05/03/2006
Postdoctoral Fellow in Watershed Hydrology, UHM C of Trop Agr & Human Res, (Manoa), Department of Natural Resources and Environmental Management, temporary for one year, renewal dependent upon performance and availability of funds. Qualified applicants are invited to send: 1) a cover letter of interest; 2) curriculum vitae, 3) transcripts; and; 4) names, addresses, fax and e-mail addresses of three referees. Application address: Dr. Ali Fares, Hydrology Lab., NREM, CTAHR, University of Hawaii at Manoa 1910 East-West Road Honolulu, HI 96822. Fax: 808-956-6539 Inquiries: Dr. Ali Fares 808-956-6361, 808-956-7530 afares@hawaii.edu .

## ID = 598, Water contents at each depth

Srilert, 05/03/2006
dear all,
I would like to ask about question of water content at each depth. My domain of simulation, I set in 1 dimension, top surface is constant flux and no flux of both side of vertical side and bottom boundary is constant pressure of water table (pressure=0). And then i varies constant flux at the surface and run for long time, i found that final water content at each depth is convergence to the constant water content . Water content at the lower depth more than upper depth, for example at 5 m. depth have water content more than at 4m. depth for each flux. and when i observed simulation of water content output, it seems water content of soil rise from water table to the upper depth. I wondering in the real situation , is it recharge from the water table or leak out from the domain? Is it correct for my simulation ? or i misunderstand in this cases.
Could any one explains me more about this simulation?
Thank you in advances.
Lert

Jirka, 06/07/2006

Lert,

If you have a groundwater table at the bottom and no flow at the top, then the water content distribution will follow exactly the retention curve and the pressure head (negative) at the surface will be equal to the hight above the water table. Thus water content will clearly be increasing toward the water table (although there is no flow). If you specify the downward flux at the surface, the pressure head at the surface decreases (more wet), and water content throughout the profile will increase. The more you increase the flux, the higher water contents you get, until you reach the flux equal to the saturated water content, when the profile will become saturated.

Jirka


## ID = 599, Example 4 – solute transport with nonlinear cation

Silvia  05/03/2006

Hello,

I have some questions about Example 4 - Solute transport with Nonlinear Cation Adsorption.

1 _ Why the hydraulic head is null for any value of the profile, while boundary and initial conditions for the flux are in the pressure head?

In Table 11.2 of the Manual there are the input parameters. One of them is "q" Darcian fluid flux density [LT-1]. Where do I specify it in Hydrus?

2 _ The adsorption isotherm coefficient, ks [M-1L3] (called "Kd" in the interface) has to be determined by inverse analysis (since "Fitted?" is selected). What does it happen when, as done in the example, minimum and maximum constraints are equal to zero?

Thank you very much for the help,

Silvia


Jirka, 05/03/2006

Silvia,

1. The experiment is fully saturated and the flow is steady state. I therefore specify constant pressure head equal to zero and fix it also on the boundary (pressure head boundary conditions). Then the flux through the column is only due to the gravity (unit gradient) and must be equal to the saturated hydraulic conductivity.

2. As explained above in (1), the flux is under these conditions equal to the saturated hydraulic conductivity.

3. When the constraints on the fitted parameter are equal to zero, that means that the problem is unconstrained.

Jirka

**ID = 600, Solute transport**

julie_an 05/04/2006
Hello,
I am very new to HYDRUS. I am trying to simulate solute transport, and i keep getting a NaN for solute mass balance results, specifically for ConcVol and cMean. Does anybody know where source of problem might be?
Thank you.
Julie

Jirka, 05/04/2006
Julie,
This means that the format (f13.3) of printing overflew. It can not overflow for any reasonable number, and thus I would assume that your entire project definition may not make sense and you are getting completely erroneous numbers.

Jirka

**ID = 601, Subserface drip irrgation 2D**

Wuqiong, 05/05/2006
hello,I am a chinese student,i want to study how to use Hydrus 2D to simulate a subserface drip irrgation process,can you help me, I always failed in run the program, can you tell me the cause,Thank you!

Jirka, 06/07/2006
Note that one of the tutorials provided at this web site deals with the subsurface drip. If you do this tutorial, it should be clear to you how to solve such problem.

Jirka

**ID = 602, Value of hCritA**

Astou, 05/07/2006
i try to simulate water infiltration in a field;i don't know what the hcritA term mean,i hope someone will answer to me quickly.

Jirka ,06/28/2006
hCritA is the minimum allowed pressure head at the soil surface. When during evaporation this pressure head is reached, the flux boundary condition (equal to the potential evaporation) is switched to the head boundary condition (equal to hCritA), and the actual evaporation is calculated.
J.

**ID = 603, Seepage Face**

mizo316, 05/10/2006
Hi Jirka -
Seepage face means that flow occurs once the boundary is saturated. Is there any specific numerical significane/advantage of using seepage face vs. other possible boundaries?

Thanks -
Mazen


Jirka, 05/10/2006
Example of a system-dependent boundary condition is a seepage face through which water leaves the saturated part of the flow domain. This boundary condition states that there is no flux across the boundary as long as the boundary is unsaturated, and that the pressure head is fixed to the zero value once saturation is reached. The flux across the boundary is then calculated from the flow field by solving the governing flow equations. This boundary condition can be used at the bottom of certain types of (finite) lysimeters, or along tile drains or the outflow part of dikes or riverbanks (Figure 22.12). In the latter case the length of the seepage face is generally not known a priori and needs to be calculated.

J.


**ID = 604, Deep drainage**

agua 05/10/2006
"Julie,
The correct function is without minus in front of Aqh:
q(h)=Aqh.EXP.(Bqh.(h-GWL0L))
You should be able to draw this function using T_Level.out file using vBot and hBot.
Jirka"
----------------------------above quoted-------------------------------------
jirka,
i am modeling 10 yr daily rainfall recharge with 30m depth (first 2m with a bit clay). the table fluctuates as rainfall recharges, so i reckon upper bc is atmospheric runoff and lower is deep drainage, so pracitcally what are the value of Aqh(meter per day) and Bqh(1/meter) approximately.
Also, what does GWL0L mean, eg. what does that mean if set GWL0L=0? or 30m?
Thanks


Jirka, 05/10/2006
Please note that there are already topics 171 and 465 dedicated to this issue.
Deep Drainage - this BC relates flow at the bottom of the soil profile to the position of the groundwater (that must be above the bottom of the soil profile, i.e., within the transport domain. There is a special function that relates this deep recharge with the position of the groundwater level. This function should predict that deep recharge is

larger when groundwater is high, and smaller when it is deep. You need to have information about this relationship before you can use it.

The discharge rate q(n) assigned to bottom node n is determined by the program as q(n)=q(h), where h is the local value of the pressure head, and q(h) is given by

q(n)=-Aqh*exp(Bgh*abs(h))

where Aqh and Bqh are empirical parameters which must be specified. This function (similarly as "free drainage") should be used only at the bottom of the profile, not sides. See the manual or the reference below.

Hopmans, J. W., and J. N. M. Stricker. 1989. Stochastic analysis of soil water regime in a watershed, J. Hydrol., 105, 57-84.

## ID = 605, Solute transport with ratios

Tine, 05/12/2006
I try to simulate solute transport under variable atmospheric conditions including evapotranspiration. I am not using concentrations but ratios as tracer input. Instead of concentrations that get higher due to evapotranspiration, these ratios are not influenced at all. Are such conditions implemented into Hydrus 1D? Do you have any suggestion how we can solve our problem? Is the source code available to change it to our experimental setup? Or is it possible to get from the water transport calculations with variable atmospheric conditions a flow field for every node and time step(here 201 nodes and 2140 days) as output-file? Thanks a lot for your help.
Tine

Jirka 05/12/2006
When water solute is applied at the atmospheric boundary, then the actual flux into the profile is calculated as
flux_Top=prec-evap.

To insure that the correct mass enters the soil profile, surface concentration is adjusted as follows:
if(Prec-rSoil.gt.0.) then
cTop=Prec/(Prec-rSoil)*cT
else
cTop=0.
end if
The same mass then enters the soil profile as required:
flux_Top*cTop=(prec-evap)*cTop=Prec*cT

Source code is available only for Version 2.0.
You can print up to 250 time levels using GUI and up to 1000 time levels if you edit directly the Selector.in file.
J.

**ID = 606, Convergence problem**

Agua, 05/14/2006
Hi,
I model the rainfall recharge using a 45m column.
Firstly i set the initial head as -100cm uniformly for free drainage and all the model works properly.
Then i switch the lower bc to deep drainage, which i think will be more real. In that case the bottom pressure head, h>0,(always merge below the table, right?) so i changed: from top to the water table level, h from -100cm to 0; from water table to the bottom, h increase from 0 to 800cm (estimated bottom is at 8m below the table), then there is a convergence problem, it came out half way during model run. so where may be the problem:
1. input daily rainfall at some point may vary too much, so it cannot converge, possible? so i increased time tolerence(up to 0.1) and pressure tolerence value(up to 10) then run, it skipped some but later still failed to converge.
2. will A,B value in deep drainage equation affect it? if yes how to change?
3. does my the initial h set properly at the graphic editor?
4. I set GWL0L=0m, make sense?
5. Min time step i tried: 1e-5, 1e-6, 1e-7. None converges.
6. I increased node number to nearly 1000, while gets denser at the two ends. Still doesn't work.

Sorry to make it looks complicated, it is an important assignment to me,
Any advice on getting rid of the convergence problem?
Many thanks


Mvcallaghan, 05/17/2006
There are so many variations in model setup, that really all I can do is offer broad suggestions. The top two problems for convergence that I have encountered are the 1) the runoff BC, and 2) high intensity rainfall. Both seem to be related to saturating the upper BC. The model does not seem to like to be saturated on its upper BC (just a thought).

Do you have "runoff bc" turned on for your surface condition? If you do, try turning it off. Do you get convergence for at least a few days? If so, look at the day when you lose convergence. Is the rainfall large? Try reducing it.
If you want your water table to be located 8 m above the base of your column, have you tried using a constant pressure lower BC?

Mike


**ID = 607, Saving Animation**

Moumita, 05/15/2006
Can we save the animation that HYDRUS-2D produces for water content (or other parameters) in a WINDOWS format (e.g., mpeg, etc.)?
Moumita Mukherjee

Mirek, 05/15/2006
No, unfortunately this is not possible. However, this function is already in our "to do" list for the new HYDRUS 2D/3D.
Mirek


Jirka, 05/15/2006
Notice that the 262 topic in this discussion forum ("Animation Results Export") provides sime guidance on how to create animations from the HYDRUS output.
Jirka


**ID = 608, Infiltration intake**

Sinishaivans, 05/17/2006
Hi,
When I run just a plain simulation in Hydrus 2-D of furrow infiltration based on soil hydraulic parameters I get the graph for the water content that I am interested in.

I would like to know which output file is going to give me the information about the infiltration intake rate (depth in a certain time) for time layers which are given in the graphical display of results? I'm asking this because I can see what is happening simply by looking at those graphs. But where is the stored data that is giving me the actual depths of infiltrated water? Which file am I suppose to look at? Graphs are apperently plotted out from some source of information. How to find it?

Any help about this I'd really appreciate.
Sinisha


Jirka, 05/17/2006
Information about where your project is stored can be found in the Project Manager.
J.S.


Sinishaivans, 06/07/2006
Dear Jirka:
Thank you for your response. However, I have checked the output files in the Project Manager as you have suggested, and I was not able to find the actual depths of infiltrated water. I can clearly see what is going on looking at the graph, but where to find that data? Can you give some more help regarding that? I'd greatly appreciate it.

Sinisha


Jirka 06/07/2006
Actual and cumulative fluxes across various boundaries are in the v_mean and cum_q.out files.
Jirka

**ID = 609, Linear constant head boundary condition**

kheatwol@mines.edu, 05/17/2006
Hi all,
I am trying to simulate infiltration of wastewater into a trench using HYDRUS-2D. I would like to simulate the bottom of the trench as a constant head boundary and then also have a vertically varying constant head boundary condition along the sidewall of the trench (where water is ponded to a certain height). Above that height on the sidewall is a no flow, as are the side boundaries. Is there any way to have a linear, spatially-varying constant head boundary in HYDRUS 2D?

Jirka, 05/18/2006
When you specify the constant head BC, in the dialog where you specify the pressure there is and check box "Equilibrium from the lowest located point". This option will take care of the changes in the hydrostatic pressure.
When you specify time-variable pressure head BC, then this is done automatically by default.

Jirka

**ID = 610, domain.in to domain.dat post-processing**

Heather, 05/18/2006
Hello,
I read a previous post entitled "How to convert "domain.in" into ASCII file" (7/10/2002) and Mirek indicated how to fix this problem in the pre-processing window. I was wondering if there is a way to convert the domain.in to domain.dat after the simultion has run?

Also, I am simulating a system with 7 regions (with 7 different soil properties) and would like to determine the maximum and minimum pressure heads within each of these regions for each print out time. Is there a simple way to do this or do I need to write a program to associate each node to each region and then figure our the summary statistics?

Finally, I was hoping that someone could describe to me the format of the Meshtria.txt file. I have figured out what some of the columns represent, but I can't seem to find any documentation to verify my suspicions.

I would really appreciate answers to any or all of these questions! Thank you!
Heather

Jirka, 05/18/2006
Heather,
a. Make a copy of the project and then save Domain.in into an ascii file. Then you will not loose the calculated results.
b. The code prints only the average pressure head for each subregion, not minimum and maximum pressures.
c. The mesthtria.txt file is described in the MESHGEN help file under the topic "Save as Text File (File menu)"

Jirka


Heather, 05/18/2006
Hi Jirka,
I read up in the manual that the h.out file gives nodal values of the pressure head at different time steps. I wanted to know if the nodal values of pressure head within this file are in sequential order, reading from left to right (i.e. nodes would be
1 2 3 4 5 6 7 9 10
11 12 13 ......etc.)
If so, then I wanted to write a fortran code to determine the maximum pressure head values within each material region at each time step, by associating the h.out file with the domain.dat file. I think that this will be possible if I can make the direct association between the two files. Do you have any suggestions?
Thanks,
Heather


Jirka, 05/19/2006
Heather,
The h.out file is printed at each "Print Time". The pressure head values are indeed sequentially for nodes from 1 to N. It should be easy to associate them with information in Domain.dat, where it is also ordered sequentially.
Jirka


Heather, 05/19/2006
Great! Thank you Jirka.
Heather


**ID = 612, Seepage at BC**

Aurélia, 05/23/2006
Hello !
This is my first HYDRUS 1D use. I wonder how to use the Seepage Face Bottom Condition. Is it just necessary to select it as bottom condition or does it need specific pressure head initial conditions (like free drainage for example) ?

Thank you very much for your answer!
Aurélia

Jirka 05/23/2006
Aurelia,
Neither Seepage Face nor Free Drainage boundary conditions require any additional parameters. Free Drainage is by default a unit gradient, while Seepage Face is by default a no flow boundary as long as the boundary is unsaturated, and zero pressure head as long as it is saturated.

Best regards,
Jirka


**ID = 613, Mass balance**

KlausR, 05/23/2006
Hi,
I am using the beta 3 version to simulate a 2D cross-sectional domain. All BCs are no flux, except for 3 constant flux nodes positioned at the middle top. The specified flux is well below the saturated conductivity of all materials. ICs are hydrostatic equilibrium with zero suction at the bottom boundary. The grid is highly refined, especially near the release nodes and near material property interfaces.

For most problems I have good MB results, well under 1% error. However, on other domains with different material properties (but the same IC and BCs) the MB is consistently about 50% error. I have double checked ICs and BCs several times, have tried constricting convergence tolerances beyond default values, and have tried setting the upper and lower limits on the interpolation tables to zero. MB errors are not greatly changed, near 50%. I have also double checked material properties for errors.

I don't understand this inconsistency and what I am doing wrong. I would appreciate any suggestions or insights.

Thanks,
Klaus


Jirka, 05/23/2006
Klaus,
To really find out what may be wrong I would need to see the project. Can you email me the project_name.h3d file?
Please note how HYDRUS reports the relative mass balance error. The relative mass balance is calculated in HYDRUS not as the absolute mass balance error divided by the mass in the domain (which is almost always very small) but as the ratio of the absolute mass balance error divided by the cumulative total fluxes across all boundaries. Thus even for very small absolute mass balance error, HYDRUS can report relatively large relative error, when the cumulative boundary flux is small (division by a small number, especialy for earlier times). Check also the Absolute mass balance error.
Jirka

KlausR, 05/23/2006
Jirka,
Thank you for the quick reply and assistance. I will email the project file.
I checked the absolute error as you suggest. It is very small relative to the total volume. Also, the specified flux is fairly small, but the cumulative flux over the entire simulation (100 yrs) is significant. Perhaps I should report absolute error relative to total volume, however, this is not satisfying given that I am generating good relative error in other domains, which also have much lower absolute errors.
Thanks

Jirka, 05/30/2006
I have looked at the mass balance problem, and found out that it is due to the precision of the stored numbers. Since I use only the real number for storing the volume of water in the entire domain (1.285616e+7), then the change in storage is below this precision. That's why it reports last mass balance errors, although there are not. I will have to convert these numbers to double precision.
J.S.

## ID = 614, Solute input

Jun, 05/24/2006
I recently got a problem:
If rainwater accumulates at the surface and the evaporation rate is very high, will the solute in the water be evaporated back into the atmosphere along with the water? Or it'll stay in the water and then increase the concentration of the remained water?
Anybody know how HYDRUS deal with this problem?

Jirka 05/29/2006
Well, I have never thought of that. Water accumulated at the soil surface should clearly become more concentrated. But I do not do such calculations. The concentration remains constant.
J.

## ID = 615, Oscillation in free drainage boundary

kheatwol@mines.edu, 05/26/2006
Hi all,
I am simulating a profile with a constant head boundary at the top and a free drainage boundary at the bottom. Output shows that the free drainage boundary flux oscillates singificantly over time. I have refined my mesh both at the constant head boundary and at the bottom and I am still getting this problem. Any suggestions?

Jirka, 05/29/2006
Lower the pressure head tolerance. J.

**ID = 616, Zn transport in soil**

Danilo, 05/29/2006
Dear all:
I have used and I'm getting use to Hydrus 2D to simulate solute transport, specifically Zn.
I am running some simulations and I'm not sure if I'm interpreting right the units, even though I consult the manuals.

The case is:
1) Water flow + solute transport in vertical plane;
2) Number of material and layers equal 4 (4 horizons);
3) Profile width equal 100 cm;
4) Units: length = cm; time=day; mass=mmol;
5) Fractional of instantaneous equilibrium = 0.75;
6) Freundlich isotherm; Beta=0.65 for all materials ; : Kd in cm3 g-1: material 1 = 3.33; material 2=2.51; material 3=1.3; material 4=1.28;
7) Alpha (kinetics sites) was estimated by EDTA extraction: material 1=0.0014 day-1; material 2=0.0085 day-1 ; material 3=0.0076 day-1; material 4=0.0106 day-1.
8) Boundary conditions for solute transport = third type
9) Domain definition: initial condition: a) concentration measured at field condition; mmol cm-3; b) sorbed concentration: ATTENTION ?: We measured total adsorbed Zn (S) but we enter the Sk as we understood that is equal to: Sk = (1-f) S.
10) It was used 10 years meteorological data (daily precipitation and evaporation);

The questions are:

1) We are looking for the amount of Zn transferred among layers (horizon) and out off profile in the simulated profile for 10 years.

2) We use some of parameters from balance.out and solute.out, however we would like to check if we interpret right the data:
a) We used ConVol in mmol cm-1 for Zn left over adsorbed at equilibrium in each subdivision (horizon) and used SorbVllm in mmol cm-1 left over at kinetic adsorbed, both from balance.out.
b) We converted values in mmol cm-1 to kg ha-1 as following:
number of mmol times 65.37 = transform to milligram Ex: 0.04 mmol = 2.6148 mg
after divided by 1000 000 to get value in Kg Ex: 2.6148 10-6 kg
So we have the unit in kg cm-1 Ex: 2.6148 10-6 kg cm-1
From that we assume this amount of Zn was in 100 cm2 (considering profile 100 cm wide and 1 cm thick) and extrapolated to 1 ha multiplying by 1000 000.
Ex: 2.6148 10-6 106 kg = 2.6148 kg ha-1 so
Kg of Zn ha-1 = mmol of Zn cm-1 * 65.37
c) Total Zn left over at subdivision (horizon) = kg ha-1 from ConVol + SorbVllm

3) We got Total Zn at initial time and subtract Total Zn at 10 years transport simulation and interpreted as the amount of Zn variation for each subdivision and for total profile.

4) We tried to cross check values as following: The balance of Zn in the domain calculated from SorbVllm (balance.out) is equal to CumchN from solute out. Is this right?

5) We also tried checking the following: The balance of Zn calculated from SorbVllm + ConVol is equal to ChemS1 from solute out?

Thank you very much,
Sincerely, Danilo

Jirka, 05/29/2006
Danilo,

9) The equilibrium sorbed concentration is calculated directly by the code from the initial liquid concentration (initial condition) and the sorption isotherm (for linear case, $s_e=f*Kd*c$). If you measured the total sorbed concentration ($s_T$) then the kinetically sorbed concentration that is entered as the input value should be $s_k=s_t-s_e$.

Mass balances:

b) It is correct that values are related to width*1 cm in perpendicular direction, i.e., 100*1=100 cm2

c) Total Zn (in the entire profile, or any subregion) = ConcVol+SorbVlIm (at any print time)

For the entire transport domain: Total Zn (final)-Total Zn (init) = sum of cumulative solute fluxes across boundaries

3) For the first layer: Total Zn (time) – Total Zn (init) = Cumulative Zn inflow (surface solute flux at time) + Cumulative Zn outflow (to lower layer at time).

For the second layer: Total Zn (Time) – Total Zn (init) = Cumulative Zn inflow (from above) + Cumulative Zn outflow (to lower layer). And so on.

4) SorbVlIm(time) - SorbVlIm(init)= CumChN(time)

5) Total Zn (final)-Total Zn (init) = sum of cumulative solute fluxes across boundaries (CumS1+CumS2+…CumS6)

I hope that this was helpful. It seems that you have all the calculations correct. Let me know if you have further questions.

Jirka

PS: Note that in our new update of HYDRUS-2D, that we call HYDRUS (2D/3D) one can define internal lines and then the code calculated actual and cumulative water and solute fluxes over these lines (e.g., from layer to layer). Thus in the new code these calculations would be done automatically by the code for you.

**ID = 617, Solute input**

Jun, 05/30/2006
Thanks for the reply.
So does that mean when the accumulated water is evaporated, the solute will be evaporated, too?

**ID = 618, Inverse Modelling**

Musagasa, 06/02/2006
Hello
I am trying to estimate soil hydraulic properties using inverse solution in HyDRUS. I intent to use water content, Runoff, Evapotranspiration and Deep drainage as my input data. The data are Time variable. Can someone explain to me what it means by "Number of data points in the objective function"? Also what do I need to fill in "number of time variable boundary records". I also face a problem on when to input my data (water content, runoff, ET, deep drainage).

Thanks in advance
Jubily

Jirka, 06/07/2006
Jubily,
"Number of data points in the objective function" - this is the number of data points which you want to uses to calibrate the model.

"number of time variable boundary records" - how many times the boundary conditions were changed.

See the existing examples (Test 2 for your second question and any inverse example for the first question).
Jirka

**ID = 619, Error with roots**

Rbeggs, 06/06/2006
I keep getting an error - "Atmospheric boundary condition must be associated with the root water uptake" when I try to run a 2D soil column with roots. The atmospheric boundary condition is set across the top and the domain data table shows a "-4" code at the top. Anyone else getting this error or have any insights on what I might be missing?

Jirka 06/06/2006
In the "Time Variable Boundary Condition" window you need to specify "Surface Area Associated with Transpiration".
J.

Rbeggs,  07/20/2006
It is not obvious how to specify a transpiration surface area for a 2-D model. The manual says that for a 2-D model, the entire computational domain is the surface, which would imply using a surface are number of 1. However, that does not seem to work properly. How is the surface area number determined for a 2-D problem?

Jirka, 07/24/2006

Transpiration is given in units of lengh/time. This value is then multiplied by the surface area associated with transpiration to get the total transpiration from the transport domain, which is then distributed over the root zone. In the older version the lengh associated with transpiration was by default equal to the surface associated with the atmospheric boundary conditions. That is OK for plants that are uniformly distributed below the soil surface, but does not work well for shrubs or trees. Thus in the new version we just let users specify whatever value they want for the surface area associated with transpiration.

Jirka


Fru, 08/29/2006

What is the unit of measuring for the time Surface Area Associated with Transpiration? is it in percentage?

Regards

fru


Jirka, 08/29/2006

It is the physical length or area. Thus it is in Length units (e.g. m) for 2D and in Surface Area units (e.g. m2) for 3D applications.

Jirka


Lassin, 10/18/2006

Hello,

I just begin in the use of Hydrus-2D. I've got the same error as rbeggs, with an additional problem : the "Time Variable Boundary Condition" command is not active in my project...


Jirka, 10/18/2006 :

"Time Variable Boundary Conditions" are active only if you specify that you have "Time-Variable Boundary Conditions" in the "Time Information" dialog window.

J.


## ID = 620, Mass balance

Jazmin, 06/07/2006

Hi,

I'd like to know if the relative error in the water mass balance is respect to the time step before, or respect to the inicial mass in the system (plus the flows)

for example, if I choose all no flux boundaries conditions for my domain

1) error(t=600) = [mass(t=600)-mass(t=599)]/mass(t=599)

or

2) error(t=600) = [mass(t=600)-mass(t=600-Dt_print)]/mass(t=600-Dt_print)

or

3) error(t=600) = [mass(t=600)-mass(t=0)]/mass(t=0)

or neither before

thanks
best regards
Jazmín
Jirka, 06/07/2006 :  19:11:30
It is related to the initial mass balance. But not in the way you describe it. See the 5.3.8 and 6.4.5 chapters of the technical manual.

Jirka


**ID = 621, Flux across internal lines**

n/a 06/09/2006
Hello,
I want to calculate the water flux and accumulated water flux across internal lines in an axisymmetric model. My objective is to determine the non-uniformity in water flux in an arbitrary horizontal plane internal to the calculation domain (not on the boundary).

On the understanding that Hydrus 2D-Standard would allow me to calculate this and present the outputs directly from the GUI I recently upgraded from Hydrus 2D with MeshGen.

Unfortunately I can not see how to achieve the above. I would appreciate any advice on this matter.
Cheers,
Andrew


Jirka 06/10/2006
Andrew,
a) When you are in the Geometry or FE-Mesh view (mode) insert Meshlines using the command Insert->Meshlines->Graphically
b) Double click on the Meshline and the dialog "Mesh-Line" will appear.
c) In this dialog, check the "Calculate Fluxes Across this Meshline". You can select up to four Meshlines.
d) Run the calculation
e) View fluxes across the meshline by selecting the command "Results->Fluxes Across Meshlines.

THis should work.
Jirka
I will try to email you an example to the email you have given here.

n/a, 06/13/2006
Jirka,
Your instructions worked well for me and the example you sent was also helpful. I have been able to calculate the fluxes I wanted.
Thank you.
Andrew


**ID = 622, Prediction of the k-function, revisited**
Hector, 06/13/2006
Dear Group,
The German Institute of Technical standards DIN has published is bringing an new version of the DIN 4220
Pedologic site assessment – Designation, classification and deduction of soil parameters normative and nominal scaling.
[Bodenkundliche Standortbeurteilung.Kennzeichnung, Klassifizierung und Ableitung von Bodenkennwerten. (normative und nominale Skalierungen)]

Herein one finds a couple of typical soil parameters expressed in the van Genuchten - Mualem relationship. Apparently the soil parameters in DIN 4220 were gained during a fitting procedure on retention data without any unsaturated hydraulic conductivity matching points. To my surprise there is a recommendation of the approach m=1-1/n for the prediction of the relative conductivity function. However experience shows, that this approach is unsatisfying with fine soils / low n-values.

If all the parameters of the van Genuchten / Mualem prediction model are empirical, as Rien stated in his posting from 07/23/2003, than one could envisage a "better" m(n) relationship for the prediction of k-functions based on psi/theta measurements for a wider range of n.

I haven't visited the parameter estimation scene for a while and would appreciate, if anyone out there could provide hints on newer approaches for predicting the hydraulic conductivity from retention data yielding more reliable estimates especially for the low n-soils.

Regards
Héctor


Jirka, 06/14/2006
Constraint m=1-1/n is needed to develop a close relationship for the unsaturated hydraulic function (van Genuchten, 1980). That is why it is almost universally used in all numerical models. For example, HYDRUS codes use this constraint as well (although I do have version where this constrain is not needed). Independent m and n parameters lead to rather complicated functions for the unsaturated hydraulic conductivity. It is included in RETC, but not in HYDRUS.
Problem with low n-soils can be overcome by using the option with the air-entry pressure head value of -2 cm (as discussed in several topics in these discussion forums).
Jirka

Rvang, 06/14/2006
Hector:
You may want to look also at a recent paper (Feb 2006) with Marcel Schaap in Vadose Zone Journal:

http://vzj.scijournals.org/cgi/reprint/5/1/27
Rien van G.


**ID = 623, Regarding the subject - heat and solute equations**

Jason, 06/13/2006
Hi everyone - The website describes that hydrus "solves the Fickian-based advection-dispersion equations for heat and solute transport". Does anyone know where I can find the heat equation and definitions of its parameters? Thanks a lot!
Jason


Jirka, 06/14/2006
In the technical manual. J.


Jason, 06/14/2006
I'm sorry Jirka, but I can't find it for the life of me. Is "Modelling Variably Saturated Flow with HYDRUS-2D" even the tech manual?
Jason


Jirka, 06/14/2006
4. HEAT TRANSPORT................................................39
4.1. Governing Heat Transport Equation .........................39
4.2. Apparent Thermal Conductivity Coefficient ..................39
4.3. Initial and Boundary Conditions ...........................40

These are the chapters in the manual:
Šimùnek, J., M. Šejna, and M. Th. van Genuchten, The HYDRUS-2D software package for simulating two-dimensional movement of water, heat, and multiple solutes in variably saturated media. Version 2.0, IGWMC - TPS - 53, International Ground Water Modeling Center, Colorado School of Mines, Golden, Colorado, 251 pp., 1999.


Jason, 06/14/2006
OK, thanks. Can I purchase this? I can't seem to find anything online.


Jirka, 06/14/2006
It can be freely downloaded from our web site together with HYDRUS-2D. I will email it to you.
J.

**ID = 624, Hydrus 1Dll**

Trenthydrus, 06/15/2006
Hi,
Is there a dll and/or source code available for HYDRUS-1D? We're trying to drive it stochastically with Goldsim...
Thanks!
trent


**ID = 625, Gravel van Genuchten parameters**

Aurélia 06/15/2006
Hello!
First of all : thanks Jirka for your precedent answer about Seepage face !!
Now I wonder if it is possible to use HYDRUS 1D for materials which have big elements such as gravels... If it is possible, how can we find examples of Van Genuchten parameters ?

Thanks for your answer ! and please say to me if my English isn't understandable...
Chat soon!
Aurélia


Jirka, 06/20/2006 :
Aurélia
I do not feel to be an expert on soil hydraulic properties of gravel materials, but since no one else answered, ...
I would either use pedotransfer predictions from Rosetta or use parameters resembling sand (a=14.5, n=2.68) and probably increase both a little bit, i.e., a=20 1/m, n=3, and measured or larke Ks. If you use larger values then simulations become unstable anyway, since you will get almost instantaneous desaturation.

Good luck
Jirka


**ID = 626, Internal lines**

Chris, 06/29/2006
Hi,
I am trying to specify a couple of internal lines (soil horizon boundaries) in a simple sloping rectangular domain. I've constructed it using the general geometry option though.
The problem I have is that the internal lines become boundary points when I make the FE mesh. This never happened in Hydrus 2D - i.e. my internal lines were just forced lines of grid points, they didn't end up as boundaries within the domain. What step am I missing here?

<<<edit>>> Just worked out that if I insert intermediate points along a line, then delete the line and make the mesh, the boundaries are as they should be (ie around the outside of the domain).>>>

Thanks very much.
Chris.


Jirka 06/29/2006
Chris,
this was one of the errors that we have fixed in Beta04 update.
See our comments on fixed errors at
http://www.pc-progress.cz/Fr_Hydrus3D_Versions.htm
Fixed error in boundary points - in 2D domains containing internal curves or more that one surface some points were marked incorrectly as boundary points

Best regards
Jirka


Chris 06/30/2006
No worries. I've downloaded beta 5.
Thanks very much Jirka.


**ID = 627, Flux from bottom to top**

Silvia, 07/03/2006
Hallo Jirka,
I would like to simulate this experiment:
- water moves from the bottom to the top of the column;
- colloid transport, with a zero gradient boundary condition at the top of the column.
It seems not possible, isn't it?
Thank you and have a nice day,
Silvia


Jirka 07/03/2006
Silvia,
No, it is not possible, since I do not have zero concentration gradient implemented for the top boundary.
But, what difference does it make if you run the experiment in the laboratory with flux upward and simulate it as a downfard flux in the model? Convection-Dispersion equation does not have gravity in it, so it should not matter how you simulate it (whether as upward or downward flow). In fact, that is how we simulated all data of Scott Bradford, who also runs it experimentally with the upward flow.

Jirka

**ID = 628, Effluent breakthrough the bottom clay liner**

agua 07/07/2006
the bottom of an effluent storage tank is at 2.5m below ground and made of clay (Ks known), given there is 5m deep effluent in it, how far is it goin to penetrate the 10cm bottom clay liner.
i have trouble model it and get the breakthrou time, coz i don't know what should be the right initial pressure head at the graphic editor. i set it as 5m along the 10cm clay column, but i couldn't see the water content breakthrough as i expect.
* where else can i tell the model the pressure is 5m above?
* i copied a example, set initial h from above to bottom as: 0, -5,-5,...-5, it works but i still don't know why should be like that?
any advice?
thank you


agua 07/07/2006
i guess initial head set as 0, -5m,-5m,... means there is a 5m pressure suck from bottom, ok i accpet that, but then why as i increase that sucking head to -7m, the water content breakthru point even delayed?!!! that makes no sense to me at all.
if i go another option, set initial condition by water content, then
1. it says error reading OBS_NODE.OUT !, any idea how come?
2. even it works i still wonder then: how i tell the model the depth of the effluent storage (5m in this case) anyway?
thanks a lot


agua 07/07/2006
could someone also tell me:
can you apply a positive pressure head as initial condition in graphic editor, say 5m at the top of a column? it seems it cannot numerically converge, is that right?

**ID = 629, Soil hydraulic model choices**

Agua, 07/11/2006
which soil hydrasulic property models is the right one to choose? any rule of thumb to engineers?
anyhow, i think different models shouldn't render big difference in result, is that right? a wierd problem: i set it in modified van Genuchten model, but the result (water conten curve) changed a lot when i changed to van G with air-entry(is what you recommand for clay materials?), anyway, after i switch back to modified van G, it doesn't converge, why?


**ID = 630, Truncation error**

Mary, 07/13/2006
I would like to know how does HYDRUS calculate the truncation and computation error produced as result of the numerical integration in both time and space. And does the HYDRUS prints these errors in the output file? RUN_INF?

I would really appreciate the help
Thank you.
Mary


Jirka, 07/13/2006
Mary,
HYDRUS expresses the calculation error using the mass balance calculations as described in the manual. This error is reported as absolute and relative value in the balance.out file.
Jirka


## ID = 631, Error message at end of HYDRUS 3D run

Brian, 07/13/2006
Good Day,
I have recently begun to use Hydrus 3D, having had some experience with Hydrus 2D in the recent past. My first model run for water and solute transport in a relatively simple system (named "Rutledge") resulted in the following error message when I attempted to access the solute results:
"An attempt was made to access C:\Program Files\PC-Progress\HYDRUS\~Hydrus3D0\Rutledge\CONC1.OUT past its end."

After pressing "OK", a second error message occurred:
"H3D_DATA.DLL has encountered a problem and needs to close. We are sorry for the inconvenience."

I am unsure as to why this is occurring. Can you please help?
Thanks,
Brian


Jirka, 07/13/2006
Brian,
I'm traveling this and next week and thus it is more difficult to resolve your problem.
I would assume that this happens because the program crashed during calculations and thus did not run till the end.
To locate the error can you try the following to see that the program is correctly installed? Can you open and run any of the examples involving solute transport without this problem. If yes, then the problem is likely in your run. Have your run your project till the end? Or did you terminate it using the ctrl+c buttons? Or did it crashed before the final time?
Jirka


Brian, 07/14/2006
Hello Jirka,

Yes, I ran an example problem which included solute transport successfully. Thus the issue is within my model. I may be having issues with setting the correct boundary conditions, and will re-build the model and try again. Thanks much for your quick response.
Brian


Brian, 07/17/2006
Good Day Jirka,
I suspect that there incorrect items in my boundary conditions that are causing the Hydrus model quickly crash. Although I am using the User Manual to guide my inputs, there must be something fundamentally incorrect in my inputs, somewhere. I am guessing that the problem is in the boundary conditions.

My model is rectilinear box. I am modeling the effect of surface distribution of a conservative tracer at the land surface on downgradient concentration. Natural gradient flow is through the xz plane in a downgradient direction. The tracer input is through downward water flux across a portion of the upper the xy plane. The upper of the model 2 m is unsaturated, and the lower 11 m is water saturated.

Instead of creating the natural gradient flow through constant head differential across the two xz planes, I am using a constant flux value at each node of the upgradient xz plane, and a seepage face at the downgradient xz plane. The constant flux value at each upgradient xz plane node is equal to the volumetric flow across the upgradient xz plane divided by the number of xz plane nodes (units: $L^3/T$). The water + tracer input flux across the upper xy plane is also input through a constant flux value that is calculated in the same manner.

The non-tracer input portion of the upper xy plane, the lower xy plane, and the yz planes are no flux boundaries.

For the conservative solute, the upgradient xz plane is a third-type boundary with 0.5 mmol concentration flux. The downgradient xz plane is a third-type boundary with no concentration value specified. The water + tracer input portion of the upper xy plane is a third-type boundary with 20 mmol concentration flux.

The non-tracer input portion of the upper xy plane, the lower xy plane, and the yz planes are no flux boundaries.

Initial conditions for water are defined using water pressure, which is negative above the water table and positive (increasing linearly) below the water table (water table value = 0 m). Initial conditions for the solute are 0.5 mmol in the entire domain.

The model is run for 5 years.

When run, the model proceeds for only about 5 seconds and then stops; it only seems to advance about 1 time step.

Are my water boundary conditions causing this to fail? I noted in the Hydrus example problems that head conditions, not constant fluxes, were often used to drive flow

across the Hydrus models. I am not doing that, but it seems that the upgradient constant flux and downgradient seepage face approach that I am using would be fine. Any troubleshooting advice would be greatly appreciated!

Cheers,
Brian


Eric, 07/19/2006
Hello,
I have also received the error messages you received. It has been a few weeks since I ran the model that gave me this messages but I seem to recall that the error messages occurred when the model either crashed or I stopped it with ctrl-break. I also recall that these error messages occurred on some models when I used ctrl-break, and didn't occur on other models when I stopped the run before the final print time. Also, the error message only occurred for the conc.out file, not for head, theta, or velocity. I know that this doesn't answer your questions about the boundary conditions in your model, but at least you know you are not alone in this issue and hopefully this will provide Jirka and Mirek w/ additional information as they refine the code.

Regarding your model setup, I only had a chance to read the description briefly but have a few thoughts. I haven't used the seepage face boundary much so I don't have much any insights regarding that, but I am curious about your specified flux boundary, which I assume is on the side of the box. How do you know what the fluxes should be (e.g., experimental data, etc.)? And, do you assign the same flux value to all nodes? If so, you may be trying to simulate an unrealistic scenario since even under conditions of a uniform horizontal gradient the horizontal fluxes through the drier regions of the unsaturated zone will be smaller than through the saturated zone. Though it appears you have avoided using a specified gradient across the domain, you might start with that 'simple' model (specified gradient and specified constant head boundaries on the sides of the domain) and then replace one boundary condition at a time until you find out which one is giving you problems. It is also possible that your model is crashing when you try to apply the tracer solution at the surface. You could test this by running the model w/ all of your other BCs intact, but without the infiltration event. Again, I only read your description briefly, so forgive me if I have misunderstood your scenario.
Good luck
Eric


Brian, 07/20/2006
Hello Eric,
Thanks for your response. I appreciate your input and advice.
I determined the volumetric water flow (Q) through the upgradient xz plane using field-derived values for saturated hydraulic conductivity (Ksat), hydraulic gradient (i), and saturated cross-sectional area (A, equal to the saturated portion of the upgradient xz plane, 11 m in thickness). The 2 m portion of the upgradient xz plane which is unsaturated is a no flux boundary. Then I divided Q by the number of nodes to obtain Q(n) values, which were used in the constant flux boundary.

The volumetric water flow Q though the upper xy plane corresponds with a surface distribution zone for pre-treated wastewater. The daily flow rate through the distribution system is known, so a constant flux boundary is used here also. The constant flux value was determined by dividing Q by the number of nodes in the distribution area to obtain Q(n) values, which were used in this constant flux boundary. The portion of the upper xy plane with no distribution system present is a no flux boundary.

Both of these constant flux values Q(n) are positive since flow is into the model. Perhaps I need to use a boundary condition other than a seepage face at the downgradient xz boundary.

I have determined the the problem is not in the solute transport portion of this model, as after I turned it off, the problem recurred.

I have been checking the HYDRUS boundary condition definitions in the Technical Manual to make certain that I am interpreting the model inputs correctly, but have not found a solution.

Jirka, do you have any ideas regarding this? I am certain that the system I am modeling is realistic, but cannot seem to get HYDRUS to run.

Thanks,
Brian


Mirek, 07/20/2006
Hello Brian,
Because Jirka is traveling this week I'll try to look at the problem. Please send me your project. I'll run calculation and after the break down I'll look into the source code in debugger what exactly happened. I do not know how large is your project - if its size is > 1MB please use WinZIP to compress it.
Which HYDRUS version do you use?
Regards Mirek


Brian, 07/20/2006
Hello Mirek,
Thanks for the response. I am using HYDRUS 3D-Standard, Version Beta.05
I have zipped the file so that it is < 1 MB and will send it to you.
Regards,
Brian


Jirka 07/24/2006
Brian
I have looked at your project. It will be very difficult to solve this problem on the course discretization (1.4 m vertically) you are using especially if you use soil hydraulic properties for sand. As you have it, you have a change in water content between the second and third node from the top from the residual to full saturation.

While you may get away with such a course discretization in the saturated zone, it will not work for the unsaturated zone. See the chapter in the manual where we discuss the spatial discretization. Spatial discretization depends on many things. It has to be fine for course materials (e.g., gravel or sand, which is your case). It has to be fine for time variable flux BC with fast changing fluxes and gradient (that is not your case). If you want to use sand, then you will need to either use much finer discretization in the unsaturated zone, or smaller initial pressure heads. Notice that the water content does not change for sands above the pressure head of about -5 m, when it is already at the level of the residual water content. Thus if you decrease your surface pressure to let's say -5 m, you will not change the water content and not cause any mass balance error, and the simulation will likely work. The infiltration into initially dry coarse-textured soils is the most difficult chalange for numerical solution. You will still need to use final vertical discretization.

Brian, 07/26/2006
Hello Jirka,
I changed made modifications to the negative pressure head as you suggested, and significantly increased the mesh density in the unsaturated zone. I also made the downgradient yz plane boundary a constant head boundary using field derived h values, and decreased the width of the model significantly since the added width did nothing that added value to the simulation.

I have one question: since the downgradient flow boundary (at the downgradient yz plane) is a constant head boundary, should the corresponding vector pointer for the solute boundary condition point to a concentration value of 0 mmol, or the background concentration in the model (0.5 mmol), or some other value? Solute flux will be exiting the model across this plane.
Thanks,
Brian

Jirka 07/27/2006
Brian,
If the model exit the transport domain, then it does not matter what type of pointer you assign for the Cauchy (solute flux) boundary. The third type boundary condition will automatically change to a second type boundary condition and solute is allowed to freely leave the transport domain. The only exception to this rule is at the atmospheric boundary, where solute is forced to stay and accumulate in the domain.
Jirka

Brian 07/27/2006
Hello Jirka,
Yes, that makes sense. However, I certainly am seeing unusual solute behavior at the downgradient yz boundary at day 365 of my model run. Solute seems to be moving upgradient across the downgradient yz plane and into the model, from where I do not know. I will send you an image of the solute ditribution at day 365. Quite strange. Do you have any hints as to why this could be occurring?
Brian

Jirka 07/28/2006
Brian,
I have looked at your image. It certainly seems that there is an inflow of water and solute into the domain at the left side. If that is the case then you should assign the pointer such, so that it uses zero concentration value. It would help to see your project. However, I'm leaving on a vacation tomorrow and thus it would be better if you send it to Mirek.
Jirka

Brian 07/31/2006
Hello Jirka and Mirek,
Thanks for your help with these issues. They have been resolved using the equilibrium pressure head profile as suggested by Jirka.

Brian

## ID = 632, Oxygen transport in soils

Ruediger 07/14/2006
Dear all,
I want to use HYDRUS2D or HYDRUS1D to model oxygen transport (at least diffusion, if possible convection) in soils depending on soil hydraulic parameters and consumption/production of microbiological processes. I know that modeling gas diffusion and recalculation of gas concentrations from solute concentrations via Henry coefficient should be possible with HYDRUS.
I would like to get into contact with persons having successfully used HYDRUS for oxygen transport modeling (or hints to respective literature)!

Thank you! Ruediger Anlauf

Jirka 07/14/2006
Ruediger ,
Both HYDRUS1D and 2D consider convection and dispersion in the liquid phase and diffusion in the gaseous phase, with the two phases linked using equilibrium Henry's law. Thus in principle it is possible to simulate O transport. The only problem may be the source/sink of O2, which at at present be simulated only using first- and zero-order processes, which can be temperature (or water content - only special version of Hydrus) dependent. I'm not really familiar if anybody actually used the code for this type of application.

I have used codes very similar to HYDRUS (their earlier versions) to simulate CO2 transport (Simunek et al., 1993, WRR; I'm on the way to Brazil and do not have here precise references), or O2 transport by Decker et al. (2006) in Vadose Zone Journal.

Best regards,
Jirka

Sascha Oswald, 10/19/2006
Ruediger,
Recently we did a preliminary study of O2 transport from the surface to the capillary fringe, using HYDRUS1D with Henry and a degradation term, related to one of our field sites. If you are still interested in the topic it may make sense to compare the simulations (approach, results, etc.).
Best regards,
Sascha


Uwe, 12/01/2006
hi,
I coupled hydrus 2d with an stationary Gas-Diffusion model for simulating O2-Diffusion:
http://home.arcor.de/hoepfner/dissertation
uwe


### ID = 633, Solute transport & Evaporation

NIMBY, 07/14/2006
Hi all,
I'm trying to simulate solute transport in an effluent irrigated orchard. My top boundary is divided into 2 subregions, both require time variable BC. For the first I used an atmospheric BC and for the second I used a variable flux BC. Now I'm trying to add solute transport and have some questions.
Is there a difference in the way Hydrus treats third type concentration BC applied on an atmospheric boundary or on a variable flux BC?
To be more specific, if I specify third type BC on the variable flux top boundary, will I have a problem since solutes will be advected with the evaporation flux? and if it does happens how do you suggest to overcome this problem?
Looking forward to your advise,
NIMBY


Jirka, 07/14/2006
Nimby,
HYDRUS treats solute flux differently at the atmospheric boundary. Obviously the convective flux at the atmospheric boundary, when representing evaporation, is associated with zero concentration, and thus all solute remains in the soil profile and accumulates at the soil surface (until it is moved back to the profile by precipitation flux).

Jirka


NIMBY, 07/14/2006
Jirka,
Thank you for your answer.

I think I wasn't clear enough. My problem is how to simulate the solute transport when having evaporation without using the atmospheric BC (since my top boundary is divided into two I can't use atmospheric BC for both).
Thanks for your help and patience,
NIMBY


Jirka, 07/15/2006
HYDRUS can consider evaporation type BC only on the atmospheric boundary. I would need to change it for you, which may be rather difficult at present since I'm on my way to Rio. If you need it now let me know and I will try to do that next week (for time-variable flux BC?).
J.


NIMBY, 07/17/2006
Jirka,
Thanks again for your help and sorry about the delayed response (long weekend).
Your suggestion seems very nice, but before I bother you, I would like to check first if using the variable flux BC makes much difference in our case.
If it creates a big error, I will ask your help again (I hope that's OK)

Enjoy Rio,
NIMBY


Jirka, 07/18/2006
If you use variable flux BC, then solute will leave the transport domain with water. If you want me to change it, send me an email so that I can do that when I return to Riverside (I will be there next Monday, the only day this summer).

Jirka


**ID = 634, Upward solute transport**

Marlin Schmidt, 07/15/2006
Hi:
I'm trying to model the upward transport of chloride from the water table into the unsaturated zone, taking root water uptake into consideration.

I'm modelling a column that's 10 m wide and 5 m deep. I've set the bottom of the column at a constant pressure head of zero. The sides are no flux boundaries. The top is an atmospheric boundary condition. I'm modelling an area where the evapotranspiration rate exceeds the precipitation, so rather than setting a precipitation rate and evapotranspiration rates, I just set an evaporation rate equal to how much it exceeds precipitation.

The goal of the model is to determine what the maximum allowable concentration at the water table can be to ensure that chloride at a criteria value never reaches the root zone.

I'm having two problems: 1) I can't see the graphs for the observation nodes. I think it may be because the output files are too big - the time runs for 100 000 days. Is there any way I

2) The chloride seems to concentrate in the root zone, so that no matter what concentration I set at the water table, it will eventually exceed the criteria in the root zone given enough time. This is the first time I've tried to model this sort of scenario, and I don't know if this is to be expected or not.

Thanks for all the help!
Marlin

Jirka, 07/21/2006
Marlin,
1. The code should display the resutls even when the file is large. It may, however, take some time to do that, since the code needs to select values to be displayed (from the large file).
2. It is to be expected.
Jirka

## ID = 635, Nodal recharge

Srilert, 07/17/2006
Dear all,
If in my simulation I would like to setup well for rechage contaminant source. Is it possible in HYDRUS 2d. I can setup nodal recharge but cannot it like a contaminant source. How should i do if i want to set contaminant source like a well in my domain? Anyone help me pls.
Thanks in advances.
Mr.Lert

Jirka, 07/21/2006
when you specify recharge in the internal node, then the cWell values (see the cBound vector) is used automatically as concentration in infiltrating water.
Jirka

## ID = 636, Switching variable head to atmospheric

Jrings, 07/17/2006
Hello,
I am using the new HYDRUS Beta, with the option to switch veriable head boundaries to athmsopheric. I am simulating a dike and gave infiltration and rain. So I

tried the switch option and noticed that the atmospheric boundary is switched to head for positive head.

Is tihis right? As I want to simulate a flooding event from one side only, but rain for both slopes of the dike, this would be unfortunate.

Maybe I did something wrong, but couldn't find out what.

Thanks for your help,
Joerg

Jirka 07/17/2006
Joerg,
The head will be applied everywhere on the atmospheric BC, where the head will be positive. If you want to have this switch only on one side, then you can specify atmospheric BC only on one side and on the other side specify time-variable head with the same flux. I hope that this will help.
Jirka

jrings 07/18/2006
I am not sure if I understand this right, I try to explain a bit more what I'm trying to do:
I have a dike, and I want to simulate flooding and rainfall. Let's assume the rainfall comes first, so then on both slopes of the dike I need athmospheric boundary conditions.
For the following flooding, on on the water-side slope I would need a variable head conditions as water may only infiltrate from there.
So i thought. If I use a head bc there and let it switch to atmospheric while the pressure head is negative, rain can infiltrate there as well; but in the flooding no water will infiltrate through the atmospheric face, which is apparently does.
I now have the idea of using two different pressure head faces for the two slopes, and only having a positive head on one of them. This should work, but only if I could choose the switch option for both variable head faces. Is this possible? The option menu only has options for variable head 1.

Thanks again,
Joerg

Jirka, 07/18/2006
Switching between atmospheric BC and head BC is implemented only for `Variable-head 1´. Why don´t you specify atmospheric BC on the water side (so that it can switch to head BC when needed) and `variable flux BC` (with the same flux value) on the other side (and thus prevent a switch to head BC on that side)?
I´m answering this during my trip to Brazil (and thus do not have much time to think about it). Greetings from Rio.
Jirka

Jrings, 10/30/2006

I have to follow up on this once more. If I do as you described, I still get infiltrating water on the variable flux side from the variable head! I even tried using Variable Ffux 2 as the only defined boundary, and still the entries from var. head 1 would create infiltration from a flooding if the "Switch from time variable head to athmospheric" was enabled, with no regard for specified flux values.

What I expect is simply that variable head 1 becomes athmospheric when this setting is chosen, and that athmospheric boundaries stay athmospheric no matter what happens with the head; and especially that flux boundaries don't have anything to do with the head (or the athmosphere).

Please correct me if I mix up things, but this is what I observed and the way I understood it.


Jirka, 10/31/2006

When you use the ""Switch from time variable head to athmospheric" option, then the code treats the `Variable-head 1´ boundary and "Atmospheric" boundary as one single boundary, and decides where to apply the atmospheric BC and where the head based on the specified head and z-coordinate.

This should not affect the other boundary conditions, such as "Variable flux 2, 3 or 4" (only 1).

If you believe that the code does not behave like that, then please send me the "project_name.h3d" file and I will have a look at it.

Jirka


## ID = 387, Flux vs sat conductivity

Tleao, 07/20/2006

hi,

I have a simple question. Having a 10 cm column, top boundary head +10 cm, and bottom = 0, the gradient is unit, right? In this case shouldn't the flux be equal to Ksat? For a clay loam, Ksat = 0.23 cm/h the resulting flux is equal to 0.46 cm/h, exactly twice. Why would that be?

thank you very much,
Tairone


Eric 07/21/2006

Hello,

The heads specified as boundary (or initial) conditions are pressure heads (h). However, hydraulic gradient is the gradient in total hydraulic head (H), and H=h+z, where z is the elevation head. So, at the bottom of your column H=0. At the top H=h+z=10+10=20cm. That yields a gradient of 20/10=2 and explains why your flux is twice what you expected.

Hope this helps.
-Eric

**ID = 638, Initial conditions problem**

Vivekgalla, 07/26/2006
hi all,
iam facing problem with the initial conditions in my problem domain , iam modelling a cylindrical column of 6 * 4 ft dimension and the soil is intially dry. my hydraulic prop are theta R = 0.141, theta sat = 0.329 alpha = 8.4, n = 1.15 ksat = 0.0209 and the boundary condition on the top of column has constant flux of 0.0001415 cubic meter / min which is equivalent to 0.003 g/min/ft2. so when i go for intial conditions tab and select for water content and input a value ( i tried for diff values ranging from 0.145 to 0.2) it is giving me error as " the initial conditions is either lower than residual wc or higher than sat wc". can anybody tell me where am i doing mistake.

and how to model for the soil whose initial conditions are dry, i.e. the initial vwc is very low.

thanks,
vivek


Eric, 07/26/2006
Hello,
I have generally had the best results when I assign the initial conditions as pressure head, rather than as moisture content. I don't know if it really matters to the model, but that has been my anecdotal experience. You might try assigning a uniform pressure head throughout your domain that would yield a moisture content slightly higher than the residual moisture content. As you have discovered, you can't use a moisture content equal to the residual moisture content. You might try a pressure that corresponds to a moisture content of 0.1415. One question is whether this pressure head (and associated moisture content) is such that there is excess water in the domain that will redistribute. To test that you can first run the model without the applied infiltration and see if the water that is initially in the domain redistributes. If redistribution is negligible, then run the infiltration simulation.
Good Luck,
Eric


Vivekgalla, 07/28/2006
hello eric,
thanks for the reply, but i am still getting problem as iam not able to see the redistribution of the flow, even after applying constant flux of 0.003g/m/ft2 at the top of column and my boundary condition at the bottom of the column is seepage face. the main problem is that i dont know what will be the head at the top of my column initially as iam starting dry and applying flux later. the flow is Axisymmetric vertical flow, i would appreciate if you have any idea for this. i would also like to know how do you get the value for pressure head for a known vwc.
thanks
vivek

Eric, 07/28/2006
Hi Vivek,
You can find the pressure head value for a known VWC by solving equation 2.18 (van Genuchten equation) in the Hydrus manual for pressure head. As mentioned in my previous post, do this for a VWC slightly greater than residual. Then, I would use the pressure you calculate as the initial condition throughout the entire column. By the way, was the infiltration into a column that was at residual moisture content, or one that was bone dry? If the former, then the approach described above may be reasonable. If the latter, I'm not sure if it would be appropriate since the actual moisture content in your system was less than the residual moisture content used in the model.

Another way to get an estimate of the pressure head that one would obtain at a low moisture content is to look at the results in the post-processing window of Hydrus. Click on the "soil hydraulic properties". The information you need is in the plot of theta vs pressure head. Right click on the graph and choose "edit data". This opens a table w/ two columns: head (absolute value) and VWC. Scroll down the table to a moisture content value slightly above residual and choose record the pressure associated w/ the moisture content. Then, you might try using that pressure (negative value) as your initial condition.

As for the fact that you are not getting any redistribution when applying flux, I'm not sure what to recommend. You might try applying a larger flux, just to see if you can get it to redistribute and make sure your model design is working correctly. If you have not done much modeling w/ Hydrus you might check the example problems. I seem to recall there are one or more problems related to column infiltration. You could even start w/ one of them and replace the hydraulic parameters, initial and boundary conditions w/ yours in a step-wise fashion.

Good luck,
Eric


### ID = 639, Cum_Q.out / Boundary.out problem

Chris, 07/26/2006
Hi,
I have a series of (2D) models set up in which I am trying to get the fluxes out of certain parts (nodes) of the free drainage boundary. To do this I am using the boundary.out file and summing the fluxes (column "Q") for particular free drainage nodes (code -6), and multiplying this by the print time as described in the Hydrus book.
However, I find that the fluxes calculated in this manner, including the overall flux for all nodes over the entire simulation, do not equate to those printed to the cum_Q.out file, which they should.
What possible reasons are there for this to occur? The model mass balance errors are all well under 1%, and the results are sensible as reported through the interface.

Thanks in advance.
Chris.

Jirka 07/26/2006
Chris,
I would assume that the difference is due to fluxes changing with time. You would probably need more print times to more precisely describe the dynamics of these changes. Otherwise, if you sum up Q values from the boundary.out file and multiply them by times differences between print times, these two values should be the same.

HYDRUS (2D/3D) can calculate fluxes accross any selected "meshline" directly. There is not need to do that the way you are doing it. These meshlines can be defined either at the boundary or inside of the domain. The code can do that for up to four meshlines. First, define the meshline(s). Second, double click on the meshline and in the dialog that appear check the checkbox "Calculate fluxes across this meshline". The code then calculates both actual and cumulative, and both water and solute fluxes across this meshline.
Jirka


### ID = 640, Problem with project manager

Gwalter, 07/27/2006
I have just installed Hydrus2d/3d. The program is installed on my C-drive which is write protected and only available to my IMS group. I have been trying to create a new project group on my D-drive. After open the NEW screen in Project Group, entering a new group name and path directory on the D-drive, and clicking OK, the new group does not appear in Project Groups. What am I doing wrong?


Mirek, 07/27/2006
Information about Project Groups is saved in file C:\WINDOWS\HYDRUS3D.ini. As your C-drive is write protected the project manager can not save your changes.
HYDRUS should have read/write access to following directories:
a/ HYDRUS installation directory + all subdirectories
b/ C:\WINDOWS\
Please contact your system administrator to solve this problem.
The file C:\WINDOWS\HYDRUS3D.ini is a text file and can be edited in a text editor (for example Notepad). If you decide to edit this file you should be careful and use the correct syntax.

Regards
Mirek


### ID = 641, Constant flow boundary conditions – simple question

Tleao, 07/28/2006
Hi,
I am trying to impose a constant downward flow trough a 1D column, by setting constant fluxes to the upper and lower boundary conditions. Therefore I am setting positive values in the upper boundary flux and negative in the lower boundary fluxes.

Is that the right way to do it. I got some misleading results, so I also tried another combinations such as ++, -- and -+ for upper and lower boundary fluxes. Any thoughts?
Thank you,
Tairone


Jirka 07/28/2006
If the flux through the column is downward, then both upper and lower fluxes must be negative (since they are against the z coordinate). The easiest way to do this is by specifying constant heads at both ends (i.e., zero) and fixing the saturated hydraulic conductivity equal to the flux.
J.


## ID = 642, Axisymmetric flow

xianghong7920, 07/29/2006
Dear all,
Are you tell me why k=2¦Ð(xi+xj+xk)/3 in The HYDRUS-2D Software Package for Simulating the Two-Dimensional Movement of Water, Heat, and Multiple Solutes in Variably-Saturated Media, p45.
Thanks!


## ID = 643, Kd and Nu in case change unit

Srilert, 07/30/2006
Dear all,
I am confused a little bit about sorption parameters in HYDRUS 2d.
The problems : I get Kd and Nu from sorption of metals onto soil test Kd=(1/10.168) litres per gram(of soil), and
Nu = 0.0074 litres per milligram(of metals)
And I got answer from last question that
Kd multiply with bulk density will get dimensionless
Nu multiply with concentration will get dimensionless
But when I set length in Hydrus I set in "metres" unit ?
I cannot find for mass unit
These I shown following are the input parameters that i input in hydrus
1.Bulk density 1500 kg/m^3 (I am not sure because I cannot find mass unit input for soils) If I correct >> so,
I input Kd=(1/10.168) m^3/kilograms(of soil) Is it correct ?
2.Concentration unit, I set in solute transport paramaters "mass unit" is "mg" , thus concentration is mg/m^3 is it correct?
therefore i get Nu= 0.0074*0.001 m^3/mg is it correct?
Could anyone please explains me more clearly?
Looking forward for yr answers? ]
Thanks in advances.
Have a nice day.
Srilert

Jirka 07/30/2006
Lert,
This is correct. Kd must have inverse units of the bulk density, i.e., L3/M. Similarly nu must have inverse units of concentration , ie., L3/M. Units for bulk densitz and concentrations can be different.
Jirka


Srilert, 07/30/2006
Dear Jirka,
Could u pls. explain more about bulk density?
I know how to set up for length unit in metre but I don't know how to change "mass unit". So, I wondering how does hydrus know the unit of bulk density if i input 1500 (mg/m^3 or kg/m^3).
Thanks you for ur answer.
Have a nice day.
Srilert


Jirka, 07/31/2006
Lert,
The best way is to look at the governing equation. Then you can see which variables multiply other variables and need to adjust units accordingly. The code does not care what units you use, as long as they are consitent. Thus since the bulk density multiplies Kd, you need to use the same units for these two variables. Similarly the variables that multiply concentrations, they need to have the same units. Note that the mass units are entered as a character (or a string) and they thus do not have any effects on calculations. They are only printed into the output and graphs.
Jirka


## ID = 644, Bulk density

Srilert, 07/31/2006
Dear Jirka,
Could u pls. explain more about bulk density?
I know how to set up for length unit in metre but I don't know how to change "mass unit". So, I wondering how does hydrus know the unit of bulk density if i input 1500 (mg/m^3 or kg/m^3).
Thanks you for ur answer.
Have a nice day.
Srilert


Jirka07/31/2006
Lert,
Since the bulk density is always mutliplied by Kd, you can use any units for these two parameters (since they cancel each other). You are not constrained by the length units specified early in the code.
Jirka

**ID = 645, Flux output meaning?**

mj 07/31/2006
I have been running yearly simulations with a time-step of 1 hour and the P-level information printing every week. The units I chose to use are cm and days.
In the NOD_INF.OUT output file, would the value of flux at each node be an instantaneous Darcian velocity in cm/day? Or would it be the average Darcian velocity for that hour (or that week)? Or would it be a cumulative value for that hour or week?

Jirka 07/31/2006
It is an instantaneous Darcian velocity in cm/day. Cumulative values (only for boundary fluxes) are in Cum_q.out.
Jirka

**ID = 646, How to eliminate crashes in batch runs**

Mvcallaghan, 07/31/2006
Hi Everyone,
Does anyone have experience programming batch runs of Hydrus 1D so that the simulation continues past a model locked in a non-convergence loop? My problem is that everytime the model enters non-convergence in a run, the batch run gets stuck in a loop that requires manual stoppage. This is not ideal as it holds up the batch run and requires manual searching for the run that crashed.

The solution would be a means of having the run terminate upon non-convergence and start the next run in the batch. Ideally an error file would be generated to note which runs did not converge. Does anyone have experience with this? Is there perhaps a kill command for the Hydrus simulation upon non-convergence? I am running tens to hundreds of simulations at a time, and I often have convergence errors now on certain extreme climate and soil conditions. It sould be great to have the batch problem solved.

Cheers,
Mike

Mirek, 08/02/2006
Mike,
I could modify the code for you so that it terminates when there is nonconvergence. However, I can not do that until the end of the August since I'm on a vacations until August 24. You would need to contact me after this date.
Jirka

Mvcallaghan, 08/17/2006
Thanks Jirka,

Enjoy your vacation! It would be be a great aid to my work if you were able to reprogram Hydrus to terminate on non-convergence. I'll contact you by e-mail at the beginning of September for an update.

I was also wondering if it would be an easy task to increase the allowable number of observation nodes to 100 instead of 10 in the source code?

Regards,
Mike

## ID = 647, Impervious bottom BC

Simony, 08/01/2006
I'm trying to simulate an infiltration process where the bottom layer is impervious. The rainfal intensity (constant 5E-6 m/s) is lower than the saturated hydraulic conductivity (5E-5 m/s), therefore I expect the upper zone to be unsaturated and the water table to build up from the bottom as soon as the bottom layer gets saturated.
I set the upper BC as "Atmospheric BC with surface run off", lower BC as constant flux=0 and "hCritA=1m".
I get very strange curves for pressure head vs time and water content vs time.
I wonder if simulating an impervious lower BC in the way I did is correct.
Thanks.

Jirka 08/01/2006
This seems correct. J.

## ID = 648, Unit of concentration

Srilert, 08/01/2006
Dear Jirka,
Thanks for your answer of bulk density.
I just to ask u another one about "Nu" parameter is

Nu*concentration = dimensionless

I change mass unit of solute is "mg" and I choose length unit in geometry information is m^3, so in concentration unit should be mg/m^3 is it right ? but in my experiment the concentration unit is "mg/L" I need to change unit from "mg/L" to "mg/m^3" ?

And then when I will input "Nu" in Hydrus 2D, I will change Nu umit in "m^3/mg" Is it right?

Could u please explain me this question causing I think it's a crucial things for my simulations in dissertation?
Thanks in advances.
Srilert

Jirka, 08/01/2006
Lert,
If you use mg/m^3 and "m^3/mg, then it is correct.
Since concentration appears in every term of the convection-dispersion equation, you are not bound by the length units selected, and you can also use mg/L and "L/mg" for concentrations and nu, respectively.

Jirka

## ID = 649, Hydrus not recognising pointer to the vector

Brian 08/05/2006
Good Day Jirka,
Hydrus is not recognizing the pointer to the vector of the boundary conditions in my model for solute transport. The initial solute concentration in the model is 0.5 g/m^3. The upgradient vertical boundary has a third-type solute BC, pointing to the -2 vector (which = 0.5 g/m^3). The model is also receiving water and solute flux from a portion of the top of the model, with a pointer to the -1 vector (which = 20 g/m^3). Water flux is created across the model via an angled pressure head.
When the model is at t=0 days, the solute concentration = 0.5 at all locations. However, after t=0 days the solute flux from the upgradient vertical boundary seems to be at a concentration = 0 g/m^3 (it should be 0.5). Meanwhile, the solute flux from the top of the model is entering the model at 20 g/m^3 (which is correct).
I cannot seem to be able to find why this is occurring. Since the model is not supposed to be receiving any water with a solute concetration of less than 0.5 g/m^3, the concentration should not be less than 0.5 g/m^3 at any location in the model, at any time.

Any help would be greatly appreciated. Thanks much.
Brian

Mirek 10/12/2006
Brian,
This problem has been fixed in the version BETA.06. In previous versions, under special circumstances, constant flux boundary conditions could be saved incorrectly. Such projects will be fixed automatically after opening in the new version BETA.06 and a warning message will be displayed (users will be asked to recalculate results).

Regards
Mirek

## ID = 650, More than one variable boundary in the simulation

DMM, 08/08/2006
I am trying to simulate vertical and lateral flow on a hillslope (2 Layers). I understand this is a 2D problem but was wondering if i can run the simulations in 1D to cut down computation time. I have atmospheric boundry condition for the top boundry.

My first question is: can I have another variable condition for the bottom of the profile?

My second question: I have run simulations with 2 layers, top boundry as atmospheric BC and the bottom as Free drainage. the top layer has a higher (308 cm/d) Ks value than the bottom layer(4 cm/day). The program keeps on crashing and would not converge. What can i do to fix this.

I appreciate all the help you can give me.

Mvcallaghan, 08/09/2006

A1. What kind of variable boundary condition do you want on the bottom boundary? I have sucessfully used a variable pressure head boundary condition to represent a fluctuating water table. It that case it is important to make sure that the function used to define the boundary pressure head is smooth. It should also be possible to use a variable flux boundary. It would require calibration to prevent excessive drying or saturation of the soil column which may lead to numerical instability.

A2. Are you using a runoff boundary condition or ponding on the upper boundary? Generally, I have found the runoff BC to be very sensitive to saturation on the upper boundary. Check your nodes just before you lose model convergence. If you are saturating your soil column, this may be your problem. I have had poor success at getting the model to converge in runoff if the upper soil column becomes fully saturated.

If you're not seeing saturation in the shallow soil, try these other ideas. Tighten you node spacing to less than 10 cm. Make sure your minimum time increment is $10^{-5}$ s at most. Remove tensioning i.e. set upper and lower tension limits to zero in Iteration Criteria menu. That's all I can think of for now.

Mike

**ID = 651, Henry's constant**

Sarah, 08/08/2006

What units does Hydrus use for Henry's constant? The help menu lists [M-1L3] and the manual lists [MT2M-1L-2]. Neither of these are units with which I am familiar.
Thanks for your help.
Sarah

Wweinig, 08/09/2006

I too have had problems figuring out the conversion from the usual Henry's Law units (atm-m^3/mole) to the dimensional form suggested in the HYDRUS help system. Any pointers would be appreciated.

Jirka, 08/10/2006
Hi,
I'm currently on vacations and do not have access to all my files.

Check the manual and look at the governing equations (definition of the gaseous concentrations). It should be pretty straightforward to figure out what the correct units are.
Greetings from Prague,
Jirka


**ID = 652, Question about tracer transport forward simulation**

Bonney,08/10/2006
Just a simple project:2 pore volumes of tracer solution, say cabr2, is injected upward to a sand column with length of 15cm, followed by 3 pore volumns of tracer-free solution. The breakthrough curve is the main point in this project. The water flow parameters are:
porosity(Qs):0.367
Hydrulic condutivity (Ks): 0.0667cm/min
Qr=0, alpha=0, n=1.1, I=0
Flow BC: constant pressure head for both upper and lower boundary.
The solute transport parameter:
time for 2pv tracer injection (pulse duration):57.17 min
Total time: 132.60 min
Bulk Density: 1.69g/cm3
Disp.=0
D_soil: 0.0335cm
all the rest are set to zero.
Solute transport BC:
Upper: concentration flux BC
lower: zero gradient
Initial concentration: 1
Profile info:
pressure heads for upper and bottom are 0. The observation point is set to the bottom.
After simulation I checked the observation points and solute transport flux. The weird things are:
1,there is a shoulder in the breakthrough curve at around 90 min and the tracer breakthrough at around 69min (but suppose to be very close to 28.5min).
2, the surface solute flux is positvie but the bottom solute flux is negative.
3, when I change the upper pressure head to 15 and lower pressure head to 0, the tracer breakthroughes at around 30min.
my questions are:
1, Is the breakthrough time controlled by the head pressure and solute transport boundary condtions? If yes, how to set them to get reasonable breakthrough curve?
2 how to set up the upward flow in hydrus-1d? Which parameter can be used to setup this condition?
3, Is there any place I can input the flow velocity in hydrus-1d?
4, How does the software calculate the flow flux?

Thanks

Jirka,08/11/2006
Quick look at your inputs:
Alpha can not be zero.
Either „Dispersivity" or „Diffusion coefficient" must be different from zero. Hydrus does not solve problems with zero dispersion coefficient.

Questions: Of course that the pressure heads affect the flux. Flux is calculated using the Darcy's law, which states that the flux is equal to the negative product of the pressure head gradient and the hydraulic conductivity.
Jirka


Bonney,08/11/2006
Thank you, Jirka. I will change the parameters you mentioned. Hope that change will eliminate the surge on BTC. Could you spend a little bit more time to answer my second question? (2, how to set up the upward flow in hydrus-1d? Which parameter can be used to setup this condition?)


Jirka,08/12/2006
Bonney,
If you want to have upward flux q and the column lenthg is L, then specify Ks=q, surface pressure 0 and the bottom pressure 2L. Linear distribution of pressure between these two values, and disable Water flow.
Jirka


**ID = 653, Rpresenting the capillary fringe**
jpjones 08/15/2006
Is it possible to set the air entry pressure to values other than -2 cm? Maybe by manually editing the files generated by the GUI before executing the model? I would like to change the air entry value to -30 cm.
Thanks,
Jon Paul Jones


Rvang, 08/16/2006
Jon Paul:
Jirka probably has to do that for you. But be forewarned that changing it to -30 cm has nothing to do with accounting for the capillary fringe as such. Alpha controls the always poorly defined extent of the capillary fringe (especially for clay soils).

See also recent VZJ paper by Marcel Schaap: http://vzj.scijournals.org/cgi/content/full/5/1/27

Rien van G.

Jirka, 08/17/2006
Jon Paul,
By manipulating the Qm parameter, you can achieved any value of the air-entry pressure head. When you specify air-entry of -2 cm, then I calculated Qm. Thus you need to select the modified VG model and then play with the Qm parameter. In the output i_Check.out (or the graph of the retention curve) you can then see what eir-entry value corresponds to selected parameters (Qm and others).

Jirka


## ID= 654, Fitting with CXTFIT

Monica,08/17/2006
Dear sir,
I am working with pesticide transport in soil columns, I am using CXTFIT to fit my BTC´s(breakthrugh curves), unfortunatelly I have found many different values if I change a little the input parameters. Firstly I tried to fit the KBR(tracer) BTC using CDE model, after that I tried to fit the parameters using CDE-MIM model to investigate the presence of two regions, I used the input parmeters (R/D= 1) and (R/D=fitted in CDE) surprisinly I observed very diferent values of R,D, omega and beta. Now I am very insecure for using CXTFIT to fitting the pesticide BTC. I would be most gratefull if you could tell me what´s going wrong.

Thank you very much for you valious attention.
Monica Milfont


Ntoride,08/17/2006
Monica:
>I used the input parmeters (R/D= 1) and (R/D=fitted in CDE) >surprisinly I observed very diferent values of R,D, omega and beta.
You seem to misunderstand D for the MIM, which is defined in Table 3.1 in the CXTFIT manual. Please also see Table 7.1 (p,266) in Bill Jury' soil physics textbook. Variance and dispersivity are defined for the CDE and MIM.

If you still have a problem, please send me an input file for CXTFIT or a project folder for STANMOD.

Nobuo
E-mail ntoride@bio.mie-u.ac.jp


Monica,08/21/2006
Dear Group,
Why in nonequilíbrium models, the programs fits for (flux concentration in equilíbrium and nonequilibrium phase)? Wich curve is valid? And why those curves in sometimes are so diferent? What is going wrong?

Ntoride,08/21/2006

Monica:

In case of the two region model (mobile-immobile model, MIM), the nonequilibrium phase is immobile. Hence, we assume only the mobile phase solution appears in effluent. The mobile and immobile concentrations are quite different when the situation is far from in equilibrium. I would recommend to carefully read Jury's soil physics textbook to understand the MIM.

Nobuo


## ID = 655, Application Error

fru 08/17/2006

"The exceptio Floating-point overflow. (0xc0000091) occured in the application at location 0x00433870."

Does anybody knows what this error means?

look forward to your replay.

Thanks

Fru


Mirek 08/17/2006

Hello Fru,

please read previous topics concerning "floating point errors" (use the "search" command and search for "floating point"). For example http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=151.

Mirek


## ID = 656, Upper and lower solute transport BC

anabdel 08/18/2006

If you simulate transport of Sodium in a soil column with atmospheric water flow upper boundary condition (with surface layer) and seepage face in the bottom. What would you choose for the upper and lower solute transport BC if you know the concentration of sodium in the ponded water (assumed constant with time) but the measurements of sodium concentration from the dripping water in the bottom is different with time (10 measurements at different times were taken). Thanks


Jirka, 08/19/2006

I would use Cauchy (or concentration flux) BC at the top and Free Drainage at the bottom.

Jirka

Anabdel, 08/21/2006
Thank you Jirka, but as you know Free Drainage is a water flow bc not a solute transport bc(and i did not select it, I selcted seepage face, but I will chose it if you recommend), so what would you choose for lower transport bc? thanks


Jirka, 08/21/2006
At the bottom of the profile, there are three boundary conditions for the solute transport. Concentration BC (Dirichlet), Concentration flux BC (Cauchy BC), and Free drainage (or unit gradient, or something like that). Select this last one. (I'm on vacations and do not have HYDRUS with me, which some of you may find surprising :-)).
Jirka


anabdel 08/21/2006
Hi Jirka,
I am sorry for bothering you while in a vacation. but the there boundary conditions for the lower solute transport bc are: Concentration BC, Concentration flux BC, and "Zero Gradient" there is no "Free drainage" or "unit gradient". Do you mean this one "Zero Gradient". Thanks


Jirka, 08/22/2006
Yes, Use Zero Gradient. J.


Anabdel, 08/22/2006
Thank you Jirka. Simulation was run and Na breakthrough curves in the soil profile were obtained. But I have noticed that NONE of the concentration curves for 14 print times (simulation period was 14 days) exceeded the initial concentration of Na in the soil (concentration at T0, which was assumed a one value along the soil profile). Do you know why? Thank you.


Jirka, 08/25/2006
Concentration in the profile can exceed the initial concentrations only if a) boundary concentration is higher than the initial concentration, b) you consider a solute source in the profile (zero order process), b) there is a root water uptake that leaves solute behind.
Jirka


**ID = 657, Using CXTFIT**

Monica 08/21/2006
Dear group,
I tried to fit other curves and I have observed that sometimes "S.E coefficient" (standart error) are bigger compared to the fitted values, ex: D=0,564 +/-39,99 (May I consider this fit as a reliable fit)? If not, whats going wrong?

Thank you very mach for you valious attention.
Monica Milfont.


ntoride 08/21/2006
Monica:
Please have a look at p. 82-83 in the CXTFIT manual.
Nobuo


## ID = 658, CXTFIT

Monica 08/21/2006
Dear nobuo,
My soil columns are saturated, the BTC's curves are simetrical so visually we can firstlly conclude that there is no two-regions. By the way I tried to fit using MIM model, and surprisingly in some cases I've got one curve (flux concentration in equilibrium phase) and the other (flux concentration in nonequilibrium phase) very diferents each other.

thank you very much for you valious attention,
Monica Milfont


## ID = 659, Pond example

Jerry, 08/22/2006
In the manual, there is an example called "pond ", however, I can't find it in the Beta05 version. Did you rescind it?
Thanks.


Mirek, 08/22/2006
The "Pond" example comes with old Hydrus-2D that can be downloaded from http://www.pc-progress.cz/Pg_Hydrus_Downloads_Form_H2D.htm . You can easily import this project into the new HYDRUS - see video tutorials http://www.pc-progress.cz/Images/Hydrus3D/TutGUI_02.htm.
Regards Mirek


Jerry, 09/01/2006
Thank you very much!


## ID = 660, Mesh Generation problem in Hydrus 2D

Vivekgalla, 08/22/2006
Hello all,
iam facing problem when i try to select automatic mesh generation for my model. its giving me error message as "Selected curve contains overlapping points (or) intersect

each other. Geometry Inconsistent!". my model is a cylindrical column with a cone shape in the bottom part and i have three differnt soil regions in my whole column with one soil in the cylinder and two diff soils in my cone part. can anyone tell how to proceed next.

thanks
vivek


Mirek, 08/22/2006
Hello Vivek,
this problem was already discussed in this forum several times. Please use the "search" command and search for "overlapping". You should get several topics, for example
http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=487 or
http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=551

Mirek


## ID = 661, Error observation point

Fru, 08/24/2006
Dear all
After run my application using Hydrus 1D an error occurs in reading the observation point. There is an error in reading the file OBS_NODE.OUT.
My observation point is positioned in the middle and to the bottom of my column, where I want to know the mass balance.
Can you someone explains me why this error occurs?
Thank you in advance.
Regards,
Fru


Jirka, 08/25/2006
Check the OBS_NODE.OUT file directly using other tools, e.g., notepade or MS Excel.
Jirka


## ID = 662, Input data

ChrisM, 08/25/2006
Hi,
a couple of questions on the input data:
1. do I have to enter the precipitation that actually reaches the soil surface (i.e. P (*minus*) EI) or the complete amount measured and corrected by the weather service?

2. do I have to enter the potential evaporation or, as my colleague told me, the pot. evapotranspiration (like HAUDE, PENMAN or PENMAN-MONTEITH / WENDLING)?

3. as I have to work on a monthly basis (means, suggesting time series to be in days), would it be better to set transpiration eq "0" (so far, I use the precipitation that reaches the soil surface by suggesting interception that depends on P and the LAI)?

Thanks a lot!

Chris

Jirka, 08/28/2006

Chris,

1. Yes. You should enter the precipitation that reaches the soil surface. The code does not consider processes, such as interception.

2. Yes. You need to enter the potential evaporation and the code then calculates the actual evaporation.

3. Transpiration is certainly more efficient in removing water from the soil profile then evaporation. Thus if you have plants that do not cover the entire soil surface, you should consider transpiration and evaporation separately. You can separate them from evaporation using LAI or surface cover.

Jirka

Fru, 10/06/2006

Sorry, I don't know anything about LAI and surface model but I need to separate evaporation and transpiration, could someone suggest me some link please?

thanks,

Fru

Jirka, 10/06/2006

You can separate evaporation (E) and transpiration (T) from evapotranspiration (ET) using the plant surface cover fraction (SCF) as follows:

T=SCF*ET and E=(1-SCF)*ET

You can observe SCF visually or you can get SCF from the leaf area index (characteristic of the plant growth) as follows:

SCF=1-exp(-a*LAI)

where a is the constant for the radiation extinction by the canopy (=0.463).

Jirka

## ID = 663, NaCl transport parameter

oliver971, 08/25/2006

Hello all,

I'm working on a model of solute transfert in andisol (a volcanic soil). We want to use NaCl but we needs the NaCl tranport parameter (Kd, Nu, Beta, etc) and Nacl diffusion coefficeint in free water.

Thanks.

Oliver971

**ID = 664, CXTFIT PVP problem**

nicvic08/29/2006
Hi,
I am trying to fit BTC data from a field experiment in order to get estimates for pore water velocity and Dispersion coefficients. I am using the Deterministic Equilibrium CDE, inverse set-up. The maximum concentration I obtained at the measurement location was about one-quarter the input tracer concentration due to mixing with other water in the system. I was able to obtain somewhat reasonable results when I entered about one-quarter of the input concentration as my input concentration. I am thinking that this may be due to the fact that I have not specified any of the constants for the exponential Production Value Problem. The CXTFIT manual cites Table 2.2 for more information about the constants, but my manual does not have a Table 2.2.
Does my problem even have anything to do with the PVP?
Any suggestions to how I can use my actual input concentration in the model, or is it just not feasible?
Thanks,
Nichole


Ntoride, 09/01/2006
Dear Nichole   F
It is generally difficult to properly estimate v and D when we do not have proper mass recovery. I am not sure what type of sink or source terms you are thinking as PVP. Solute transport may not be
one-dimenstional and you might observe concentrations at limited flow paths. This may be a reason for the small observed concentration.
Nobuo

I noticed the U.S. Salinity lab web site is not available to download the CXTFIT manual. If you want, I will send you a pdf file.


Wendylcy, 09/01/2006
Dear ntoride:
Nice to see you in the forums,l cannot downlord the CXTFIT manual. can you send me the file?


Ntoride 09/01/2006
We will put the CXTFIT manual in this PC-progress site soon. We will let you know when it becomes available. Nobuo


Rvang, 09/02/2006
Nobuo, and others:
The CXTFIT manual and other manuals/programs can be downloaded from
http://www.ars.usda.gov/Services/docs.htm?docid=8910
which is the new dedicated USDA-ARS site. For STANMOD and HYDRUS, users are routed also to the hydrus site (www.hydrus2d.com) for the latest versions. If

needed, send an email to Walt Russell (wrussell@ussl.ars.usda.gov) to get a copy of the CD with all programs and manuals, plus other related material/publications.

Hope this helps. --Rien van G.

Mirek, 09/03/2006
1. Now all manuals for original STANMOD programs (i.e. 3DADE.PDF, CFITIM.PDF, CFITM.PDF, CXTFIT.PDF, CHAIN.PDF and N3DADE.PDF) can be downloaded from our WEB site at http://www.pc-progress.cz/Pg_Hydrus_Downloads_Form_SAM.htm

2. We have updated STANMOD installation (StAnMod_24.exe) which now contains all original STANMOD programs (listed above). These programs are installed into the "Original STANMOD Programs" subdirectory of the STANMOD installation directory.

Regards Mirek

## ID = 665, Kinematic wave problem

Ekkehart08/29/2006
Dear Jirka,
we are trying to fit an HYDRUS simulation to our experimental data taken from field infiltration experiments. We assuming, that the difference between the simulated and observed fluxes is caused by the macroscopic poresystem in our soil and would like to simulate the water flux in these pores with an kinematic wave model. The dual-permeability model MACRO can not handle pressure head information for the upper boundary condition. In one of the last topics you have written, that in former hydrus version you have implemented dual-porosity models. Is there also an older version which takes into account dual-permeability or kinematic wave approximations. If yes, do you think it is possible to use on of this older versions for our purpose?
Ekkehart Bethge

## ID = 666, Groundwater flow through boundary

Jerry - 09/01/2006
I just recently started to use Hydrus 2D/3D and I have a question regarding how to setup a boundary to reflect the real situation.

I am trying to modeling the impact of infiltration from a pond to the groundwater. I setup a domain of 15 m deep and 5 meter wide. I specified the groundwater level to be 2 meter above the bottom of the domain, 2 meter of aquifer in the domain. I specified constant flux from top of the domain, and my question is regarding the bottom part of the domain.

I specified constant head equilibrium with the depth along the boundary that is in groundwater. I set one side of the boundary lower than the other side, so that it may

flow through. And I know the groundwater is flowing 100 ft/year. My question is, is this right way to specify this condition? How do I get the flux coming out of the groundwater boundary only? The results shows the constant boundary flux, which seems combining the constant flux on top of the domain also.


Jirka, 09/01/2006
Jerry,
HYDRUS indeed sums up fluxes across all constant flux and head boundaries and thus in your set up you would get only one flux across all boundaries. If you want to have fluxes reported for different part of boundaries, there are two ways of doing it:

a) Using Mesh-lines. You can specify a mesh-line for each part of the boundary. Once you enter the mesh-line and double click on it, you will get a dialog where you should check "Calculate Fluxes across this mesh-line".
b) Using different type of boundary conditions. Instead of using a constant head at the sides of the domain, you can specify a variable head BC 1 and 2 at the left and right side, respectively. The reported fluxes will be for each boundary separately.

I hope that this helps
Jirka


Jerry09/01/2006
Thanks Jirka, the mesh-line is really handy.


**ID = 667, H mean value too high**

Fru,  09/01/2006
Hi All
I am simulating drainage with root uptake within a domain (150 cm) which has a top atmospheric BC with surface runoff and on the bottom free drainage. There are four different materials in my domain but I just want to know the mass balance in a single sub region. As initial condition I'm using soil moisture data and one year records of time variable BC.
The resultant must balance information I have seams to be strange, especially the h mean value seams to be too deep.
Could you please to have a look to my results below and tell me what do you think are the mistakes I'm doing?

Sub-region num. 1
-----------------------------------------------------------
Area [L] 0.15000E+03 0.15000E+03
W-volume [L] 0.24105E+02 0.24105E+02
In-flow [L/T] -0.63907E-01 -0.63907E-01
h Mean [L] -0.15641E+05 -0.15641E+05
Top Flux [L/T] 0.28580E-01
Bot Flux [L/T] 0.40011E-04
WatBalT [L] 0.66895E-01

WatBalR [%] 0.625
-----------------------------------------------------------
As I know which is the GWL (-350 cm) I would like to insert the value but could someone explain me why in time variable boundary condition window I'm missing parameters like rGWL, GWL and…?
Thank you in advance,
Regards
fru


Jirka 09/01/2006
fru,
1. I can see that you are using HYDRUS-1D. Please, ask questions about HYDRUS-1D in the HYDRUS-1D discussion forum and not in HYDRUS-2D.

2. I do not know what to say about your mass balance. I guess your transpiration is much large than precipitation and the profile is thus dry. I can see that the average water content is about 0.16 which must correspond to the mean pressure head you have. Check your retention curve whether it is true.

3. Free drainage bottom BC does not need any additional values, and therefore in the Time-variable BC dialog window you are not asked to enter rGWL (time-variable bottom flux) or GWL (time-variable bottom pressure head). You would be asked to fill in this information, if you selected these bottom boundary conditions.

Jirka


### ID = 668, The output file of concentrations in profile

Anabdel,09/05/2006
Modeling friends,
For a transport simulation in Hydrus 1D, where do you find the out put file that contains the simulated concentrations of a solute (two solutes in my case) with time (14 days here) in the soil profile (depth). I have selected the out put concentrations to be printed every day.
Thank you.


Jirka, - 09/06/2006
In the NOd_inf.out file.
J.


Anabdel, 09/07/2006
Thank you Jirka but I only see in that file data for one solute concentration at the 4 observation points i selected (the last 4 columns to the right labeled "Conc(1...NS)". I don't see data for two solutes. Shouldn't I get two solute concentration data times 4 observation points? Thanks

 Anabde, l09/08/2006
Thank you Jirka but I only see in that file data for one solute concentration at the 4 observation points I selected (the last 4 columns to the right labeled "Conc(1...NS)". I don't see data for two solutes. Shouldn't I get two solute concentration data times 4 observation points? Thanks


Jirka, 09/09/2006
Anabel,
First of all, you can view all these results in the GUI where it should be very clear what is what. Anyway, you see at the end of the table four columns likely because you consider nonequilibrium transport and then the code reports not only two concentrations in the liquid phase (two solute concentrations), but also two concentrations in the nonequilibrium phase (either concentrations in the immobile water or sorbed concentrations at kinetic sites, depending on options you chose).
Jirka


## ID = 669, Vertical flow engineeered wetland

Brian, 09/05/2006
I have a couple of questions regarding a 2D vertical flow (downward), saturated model I have created in Hydrus. I am attempting to model a pulse of a conservative tracer in a vertical flow, saturated engineered wetland. I would like to compare the Hydrus-generated tracer breakthrough curve with a suite of conservative tracer breakthrough curves which are generated using a continuously stirred tank reactor (CSTR) tanks in series equation, for n tanks ranging from 1 to 5 in increments of 0.25. I am seeking to determine how many tanks in series best represents the engineered wetland.

The Hydrus model has dimensions x = 100 cm, z = 150 cm (the thickness of the engineered wetland). Based on the water loading to the engineered wetland (EW), I have calculated q = 0.00094 cm/sec. With n = 0.45, v = q/n = .0021 cm/sec. The hydraulic gradient from the top of the model to the bottom has a design value = -0.003. I have used constant head boundaries at the top (150 cm) and bottom (145 cm) of the Hydrus model which are not realistic, but which do result in the correct hydraulic gradient: dH/dL = (145 cm – 150 cm/150 cm – 0 cm) = i = -0.003. Based on this gradient, n = 0.45 and the v calculated above, and using v = -Ki/n, I obtain Ksat = 0.3125 cm/sec, which is reasonable for the grain size of the material in the EW (i.e., gravel).

Based on z = 150 cm and v = 0.0021 cm/sec, the hydraulic residence time = (150 cm)/(0.0021 cm/sec) = 72,012 sec = 19.8 hours. This is the design hydraulic residence time (HRT) in the EW being modeled, and is realistic for this system.

The solute tracer is input with normalized conc. = 1 for the first 6000 sec of the model run. When I ran this model, the tracer had exited the model well before 1 HRT was reached, which is unreasonable. Also, the downward velocity in the model was 0.3 cm/sec, much greater than the desired v = 0.0021 cm/sec. I tricked the model into reducing v by decreasing Ksat to 0.002 cm/sec. This did indeed reduce v to the

desired v = 0.0021 cm/sec, but the centroid of the solute pulse was now exiting the model at 0.5 HRT, not 1 HRT. Strange.
A few questions:

1. I suspect that the effect of gravity in a vertical flow, saturated system must be modeled differently than how I have proceeded. Darcy's Law shows that for a saturated, vertical flow system, q = - (permeability/viscosity)*[(dp/dl)-(density*gravity)], where dp = change in pressure head and dl = length of system. I am not quite sure if I am using this idea correctly in my model. How should the vertical flow condition be modeled?
2. Could I use a seepage face to construct the boundary conditions? How? Is there a better set of boundary conditions to use for this particular application?
3. For a pulse input of conservative tracer, what is the appropriate amount of time to input the tracer into the system?

Thanks much,
Brian


Jirka, 09/07/2006
Brian,
1) The problem is that you do not consider in your calculations gravity.
You have:
v = 0.0021 cm/s
q = 0.00094 cm/s
Gradient G = -0.003
Ks=-q/G=0.00094/0.003=0.3125 cm/s
H_bot=150 cm

Then
If the flux is supposed to be downward, than G must be positive (flux negative, since z is positive upward)
G = dh/dz + 1 = (h_top – h_bot)/dz + 1 => h_top = (G - 1) dz + h_bot = (0.003 - 1) 150 + 150 = 0.45 cm
Thus to have a gradient of 0.003, you need to have the top pressure head equal to 0.45 cm when the bottom pressure head is 150 cm.

2) You can not use seepage face with such a small gradient, since then most of the profile would necessarily be unsaturated.
3) I do not understand this part of the question.
Jirka


Brian, 09/07/2006
Hello Jirka,
Thanks for the explanation. I understand.
After continuing the think about the issue, and considering the true design of the vertical flow engineered wetland, I redesigned the model similar to the Wetland 2 example included with HYDRUS. The model now includes a pipe at the bottom connected to a standpipe at the side (similar to the true design), which is set 5 cm

below the top of the vertical flow engineered wetland. I used a constant flux BC at the top of the wetland, corresponding to the linear loading rate of water, and a seepage face at the top of the standpipe. Instead of a drainpipe I made the underdrain and standpipe a different material, with porosity = 1 and a very large hydraulic conductivity (mimicing a pipe). Using the input variables previously described, I was able to model a conservative solute pulse throught the wetland that had a modeled residence time equal to the design residence time.

Regarding the pulse input, let me re-word the question: what is the best way to model a Dirac pulse into my model? Say the model is to run for 1 day. Should the pulse of conservative solute be input into the model domain for the first 600 seconds, or 1200 seconds, or 6000 seconds, etc.? A step input of tracer is easy to model, yes, but what about a pulse? Does it really matter how long the pulse lasts? Is there a rule of thumb for the proportion of time that the pulse should be input, relative to the total model run time?
Thanks,
Brian


Jirka, 09/07/2006
Brian,
The only think important is that the duration of the pulse is relatively small compared to the residence time. Also the numerical time step (at least initially during the pulse) should be smaller than the pulse. I would probably suggested to use pulse of 600 s, and initial time step of 10 s. Just make sure that the right mass (i.e., conc*flux) enters the domain.
Jirka


**ID = 670, Constructed wetland modlule error message**

Brian,09/06/2006 : 18:34:24
I am encountering an error message when attempting to run the Hydrus Constructed Wetland (CW) module. The error message in the DOS shell window is as follows:
Welcome to HYDRUS
Vertical plane flow, V = L*L
reading nodal information
reading element information
reading geometric information
reading material information
generating materials
reading time information
reading seepage face information
reading solute transport information
Error when writing to an output file!

I will admit that this is my first time using the CW module, and I am just getting familiar with it. Has anyone else encoutered this error message?
Thanks,
Brian

Jirka, 09/07/2006
Brian,
Please, send me the project_file.h3d file. Thanks.
Jirka


## ID = 671, Particle (500nm) transport

Sharendr, 09/07/2006
Hi
I want to model particle ( iron partilces( around 500 nm)) transport in soil media. What software do you recomeend me to use? Is it STANMOD or Hydrus 2D or infiltration model in soil media. Please recommend the best software to do the modelling.
Hari


Jirka, 09/07/2006
harendra,
If the flow is under steady-state conditions and fully saturated then you can use both STANMOD or HYDRUS. HYDRUS has the attachment/detachment model that is usually used to simulate transport of particles (viruses, colloids, bacteria, ,,,) implemented directly. CXTFIT in STANMOD uses the two-site sorption model. Using simple mathematical manipulation, you can show that this model is mathematically identical to the attachment/detachment model and thus you can use it as well.
Jirka


 Sharendr, 09/12/2006
Thanks a lot Jirka.
harendra


## ID = 672, Particle transport (500nm)

Sharendr, 09/07/2006
Hi
I want to model particle ( iron partilces( around 500 nm)) transport in soil media. What software do you recomeend me to use? Is it STANMOD or Hydrus 2D or infiltration model in soil media. Please recommend the best software to do the modelling.

harendra


Jirka, 09/07/2006
harendra,
First, please do not ask the same questions in different discussion forums on our web site!

If the flow is under steady-state conditions and fully saturated then you can use both STANMOD or HYDRUS. HYDRUS has the attachment/detachment model that is usually used to simulate transport of particles (viruses, colloids, bacteria, ,,,) implemented directly. CXTFIT in STANMOD uses the two-site sorption model. Using simple mathematical manipulation, you can show that this model is mathematically identical to the attachment/detachment model and thus you can use it as well.

Jirka


**ID = 673, partilce transport (500 nm)**

Sharendr, 09/07/2006
Hi,
I want to model particle ( iron partilces( around 500 nm)) transport in soil media. What software do you recomeend me to use? Is it STANMOD or Hydrus 2D or infiltration model in soil media. Please recommend the best software to do the modelling.
harendra


**ID = 674, Water balance**

Fru, 09/07/2006
Hi Jirka
As I want to know how much water gets in the garbage (located under my profile, under my free drainage bottom BC) after evaporation and plants uptake, I'm wandering if this amount of water is given by the difference of Volume (V1-V2) or by the single Inflow value. I know there was some discussion on that topic already in this discussion forum but I'm not sure I understood well.
And also why in the Hydrus 2d there are not values like Top/Bottom flux?

Thank you for your time and help
Regards,
fru


Jirka, 09/07/2006
Why the Hydrus 2d code does not have values like Top/Bottom flux:
HYDRUS-2D offers you several types of boundary conditions (constant or variable head or flux, free drainage, deep drainage, seepage face, atmospheric, ...), but you can specify some of them (head or flux) on any part of the domain, both top or bottom. Therefore HYDRUS reports fluxes only for different types of BCs and users should remember where they specified these boundary condtions.

Thus if you want to know what flux goes from the bottom, you need to look at particular BC assigned at the bottom.

HYDRUS-2D integrates or fluxes for a particular type of BC. Thus you should not specify the same type of BC at different parts of the boundary, if you want to have individual fluxes. In HYDRUS (2D/3D) you can specify so called Mesh-Lines and evaluate flux across any cross-section through the domain or part of the boundary.
Jirka

Fru, 09/08/2006
I'm sorry Jirka but I think I haven't been clear in the last message.
I don't want to know the flux from the bottom I want to know how much water from the top arrives to the bottom (after evaporation and plants uptake).
I will explain my work better:
I am simulating drainage with root uptake within a domain (150 cm) which has a top atmospheric BC with surface runoff and on the bottom free drainage. Under the 150 cm of profile there is garbage (so the garbage is not in my simulation) and I want to know how much water arrives in the garbage between two times. There are different materials in my domain but I'm just interesting to know the water balance on the bottom, so I choose a single sub region.
As initial condition I'm using soil moisture data and one year records of time variable BC.
1. This amount of water in given from the inflow value or from the difference in water volume (V2-V1)?
2. Why do I need to specify different BC for different part of the boundary?
3. In Hydrus 1D I choose exactly the same BC as Hydrus 2D (Top: Atmospheric with surface run off, Bottom: Free drainage) but in the 2D version the mass balance file doesn't gives to me value like top/bottom flux, why?
Regards,
Fru

Jirka, 09/09/2006
Why do not you look for fluxes for the Atmospheric BC and Free Drainage BC? Those will likely be fluxes across the top and bottom boundary, respectively.

## ID = 675, Hydrus 1D help on CO2 parameters

Sjones, 09/09/2006
Jirka,
Is there an update to the H-1D help files for CO2 parameters or is there a publication where these are defined? I'm looking for dimensions and examples of some of these parameters.
Thanks, Scott

Jirka, 09/10/2006
Scott,
The CO2 module is in detail described in these two papers. The second paper presents a discussion on majority of CO2 transport and production parameters.
Jirka

Šimùnek, J., and D. L. Suarez, Modeling of carbon dioxide transport and production in soil: 1. Model development, Water Resour. Res., 29(2), 487-497, 1993.

Suarez, D. L., and J. Šimùnek, Modeling of carbon dioxide transport and production in soil: 2. Parameter selection, sensitivity analysis and comparison of model predictions to field data, Water Resour. Res., 29(2), 499-513, 1993

## ID = 677, Concentration at water table

Darren, 09/15/2006
Hello all,
I'm simulating transport through the vadose zone in h1d and am interested in the concentration once in the saturated zone. I am assuming the "cgwl" will calculate the concentration of the saturated region. In the model, I have unsaturated soils above a layer of full saturation. However, the model will not converge under this scenario. I then tried placing only the bottom boundary as a constant pressure at saturation, and the model converges but the "cgwl" is all zeroes. Recommendations would be greatly appreciated. Thank you.

Darren

Jirka, 09/15/2006
Darren,
cGWL is the average solute concentration in the saturated zone (thus you do need to have saturation zone to get this). If you specify zero pressure head at the bottom of the profile, then there is no saturation zone in the profile. In that case you can use cBot as the bottom concentration, or you can place an observation node at the bottom.

There is no reason why the code should not converge when there is the saturation zone within the domain (except that, as often happens, you fource so much water in, that the entire profile becomes saturated). Perhaps relax the pressure head tolerance to 1-3 cm.

Jirka

## ID = 678, CO2 diffusion

Marina Gillon, 09/15/2006
Good morning,
I use Hydrus 1D version 3.00 for $CO_2$ diffusion, and I have several questions :
1. The input diffusion coefficient of $CO_2$ in carbon dioxide transport parameters is a free air diffusion coefficient or the soil diffusion coefficient?
2. In the hydrus 1D manual, the $CO_2$ production terms f(hf), f(t) for root and bacterial productions and f(x) for root production are not detailed. What are formulas of these production functions ?

3. I would like to use the option of $CO_2$ diffusion for the diffusion of isotope $13CO_2$. Is it possible?

Thank you for your answers
Marina


Jirka, 09/15/2006
Marina,
a) This should be the free air diffusion coefficient. This is further multiplied in the code by the tortuosity coefficient to get the soil dissufion coefficient.

b) f(hf), f(t), and f(x) for root production correspond with the root distribution, as specified in the Root Water Uptake part of the code (i.e., Feddes reduction function or S-Shape function for f(hf), root growth for f(t), and root growth or root distribution for f(x).
f(h) for bacteria production is given by 5.20. There is no f(t) for bacteria prodiction (see page 63 of the manual)

c) I would expect so, but I do not have any experience with this, so do not take my word for this.

The CO2 module (and its reduction function and coefficients values) is described in detail in two publications:

Šimùnek, J., and D. L. Suarez, Modeling of carbon dioxide transport and production in soil: 1. Model development, Water Resour. Res., 29(2), 487-497, 1993.

Suarez, D. L., and J. Šimùnek, Modeling of carbon dioxide transport and production in soil: 2. Parameter selection, sensitivity analysis and comparison of model predictions to field data, Water Resour. Res., 29(2), 499-513, 1993.

Jirka


**ID = 679, Multiple runs**

Bruno, 09/15/2006
Dear all,
I'm making multiple runs of hydrus under the control of an external model, so i would like to avoid having to press "enter" each time HYDRUS finishes its work. I've read some on the subject in the F.A.Q. section (part 27) as well as in an older topic, but it stays quite unclear to me nevertheless...

As i'm dealing with inverse problems only, the command line i'm using is simply :
c:\hydrus1d\h1d_clci.exe

From what i saw, i'm supposed to change it into :
c:\hydrus1d\h1d_clci.exe<return.txt

I know little about DOS commands, but the symbol < must be there to redirect the awaited instruction from the keyboard to the "return.txt" file. As this instruction is "press enter to continue", it should appear in a different way in "return.txt".
So, could anyone help me with the content of this text file ?

Best regards
Bruno


Jirka, 09/15/2006
Bruno,
In version 3.0 of HYDRUS-1D you can disable the need to hit the Enter button simple on the input. See the Print Information dialog window (the check box "Hit Enter at the end?)".
You can also open and run multiple projects simultaneously to avoid these DOS commands.

Jirka


Bruno, 09/17/2006
Jirka, thank you for your quick reply.
I'm using Hydrus 3.0 and the "Hit Enter at the end" option had been previously disabled. I've also checked out that the "lEnter" variable had the correct "false" logical value in file "selector.in"
But the problem remains the same... I should maybe tell you what i'm doing exactly to help you or anyone else figuring out a solution :

- Hydrus is given a set of parameters, whose values are written in "selector.in" and "fit.in"
- Hydrus is allowed a few iterations for adjustment of these parameters
- Results are compared with disposable observations, reading data recorded in the file "fit.out"
- A new set of parameters is proposed, and so on...

No error messages appear during the process, but automation seems impossible due to something in Hydrus,
as the parameter estimation process i'm using has been successfully tested on (little) control programs.

Shall i have to use the trick with the file "return.txt" containing an enter sign, and would you in this case explain me exactly what to do ?

Best regards and a nice sunday !
Bruno


 Jirka, 09/17/2006 :
"Hit Enter at the end" should be enambled in version 3.0. lEnter equal to .false. in the selector.in file should result in the code closing the DOS window when it is finished;

but only for direct simulations (if it is disabled, download HYDRUS-1D again). This has not been meant to work for inverse simulations. For inverse simulations you need to use that trick discused here somewhere in the discussion forum.

I could either send you an example of such file, or modify the code so that lEnter works also for the inverse problem. However, I can not do it now since I'm on my way to China. If you need it contact me after September 27 when I come back.

Jirka


mvcallaghan09/18/2006
Hi Bruno,
I do alot of batch runs with Hydrus. I needed to create three files. The first was a batch file, which I simply called run.bat. The run.bat file contains the following text:

copy Path1.txt level_01.dir
H1d_calc<return.txt
copy Path2.txt level_01.dir
H1d_calc<return.txt
...etc.

,where Path1.txt, Path2.txt are the second aforementioned files, and return.txt is the third file. All three files are located in the hydrus1d program directory.

The contents of Path1.txt (Path2.txt, etc.) is:

C:\Water Balance\Park0001
,which is an example location of the hydrus source/run files.

The contents of return.txt is:
<0D0A>
,which is the hexadecimal? code for the return keystroke.

The run.bat file can have as many run paths as you want, and each path must be specified in a PathX.txt file.

To run the batch, simply click on the run.bat file in Windows. The DOS shell should start up automatically.

Good luck,
Mike


Bruno, 09/19/2006
Hi whoever goes by the name of mvcallaghan, and thanks for your advice :
i'll keep it in mind for some future occasions,
while i've found a slightly different solution by myself.

concerning the 'return keystroke' problem, instead of typing <0D0A> in return.txt,

i just pressed the return key and that was it, return.txt is apparently composed of
a blank line and in fact of an 'enter sign'...
so the instruction c:\hydrus1d\h1d_clci<return.txt works the way it should.

now about the necessity to create the level_01.dir file :
i sometimes had the corresponding error message, and sometimes not,
could it be related to manipulations made in hydrus input/output files ?

couldn't it be the result of some sort of a 'confusion' in hydrus ?
for example in such a situation : on one hand you run hydrus through
the windows interface and on the other hand you change parameters
directly in input files, and then back again, both processes being active at the same
time.

now my batch runs don't need the set of instructions you describe,
and they've been built just after a proper re-definition of the problem,
with newly created input files.

waiting to read from you
thanks again
Bruno


## ID = 680, Variable Pressure Boundary

m_north, 09/20/2006
Dear all,
I am trying to simulate the wetting of a hydrologic margin adjacent to a lake in
Antarctica. Therefore, I have the cross-section of a wedge (I guess somewhat
triangular) that begins at the water's edge and extends into the dry soil zone away
from the water. I have specified a constant head boundary at the bottom of the model
near the stream as a water table with h=0. Then (also along the bottom of the model,
but further out) there is a no-flow boundary extending from this representing the
permafrost. ie the water table grades in to the permafrost.

I have questions about:
1. How to specify the top boundary.
2. Initial Conditions

1. At first I specified the top as an atmospheric boundary. However it would not
converge. When I specified it as a variable pressure boundary it will converge and
gives me results as I expect from field measurements. (wet near the water and dry far
from the water). The trouble is that I understand I need to specify GWL values
through time if I put a var press bound. I have one value as 0 currently. (I don't really
understand the GWL value. Is the value you input the head value that is used for
every node in the model? I don't want to specify heads through time b/c I rather want
to see how the model comes up with heads through space). Secondly, I wonder if the
precip and evap data are not being used anymore b/c I have replaced the atmospheric
boundary with a var press bound.

2. I began by running the model completely dry except for the bottom constant head boundary where the water table is. This gives very poor results. If I linearly distribute heads from h=0 at the bottom to h=-80 at the top of the model, I get much better results. However, if I change the initial conditions (ie change the top head a little) I get varying results. How can I get a more stable solution?

Thank you so much for your assistance!
Melissa


Jirka, 09/20/2006
Melissa,
there is quite a lot of questions. I will try to answer some.
1. If you do not use atmospheric BC then the precip and evapor values are not used. You should use atmospheric BC as the surface BC and predict pressure heads and compare them with your measurements. Why it does not converge with atmosph. BC? It is difficult to figure out why since there may be a lot of that may be specified wrong to cause nonconvergence. We gave throughout this discussion forum many suggestions how to solve this.
2. To have profile completely dry with specified bottom boundary head does not make much sense, since the water start rushing upward from the BC to bring the profile to equilibrium. To have other profile, such as linear change with depth or even hydrostatic equilibrium make much more sense.

J.


m_north 09/21/2006
Dear Jirka,
Thank you for your helpful response. I have been trying to get the simulations to work using the atmospheric boundary at the top; they will converge now, but the results are all wrong. I'm working on it. It is sad that the pressure head boundary which worked so perfectly is wrong! The problem must then be with the evap and precip input values.

Thanks again,
Melissa


**ID = 681, Temperature data**

Bruno, 09/21/2006
Hi everyone !
Having solved problems in batch runs with your kind help (see topic "multiple runs")
i return with some questions this time on Heat Transport in Hydrus.

Suppose you use upper and lower boundary conditions in terms of variable temperature data,
as listed in the "Heat Transport Boundary Conditions" window.

Next step, you have to fill in the "tTop" and "tBot" columns in the "Time Variable BC" window.

This once made, the "Ampl" column still needs to be considered, whereas the amplitude
of the temperature variation is already (implicitly) present in the "tTop" data.

Thus informations in the "Ampl" seem to be redundant, as are those in the "Heat Transport Parameters"
window, where the user is asked for "temperature amplitude" and "interval for one temp. cycle"

Dealing with sine variations (over the day or over a year) i guess i obtain relevant results when
selecting a 0°C amplitude and giving the right value for the temperature cycle.

And what if the temperature variation is not a sine phenomenon ? Shall i put 0 again for both
amplitude values in the two different windows ?

Best regards.
Bruno


Jirka, 09/21/2006
Bruno,
The "Ampl" from the "Heat Transport Parameters" dialog is used when there are no time-variable data entered. When such data are entered, then the "Ampl" from the "Heat Transport Parameters" dialog is overwritten with the "Ampl" from the "Time Variable BC" dialog.

Obviously if you do not want to use this "Ampl" option, you can specify it equal to zero and describe the actual variation in temperatures using the information entered in the "Time Variable BC" dialog.

Jirka


**ID = 682, Sorption and initial concentrations**

Darren, 09/23/2006
Hello,
When simulating a sorbing solute in H1D, the initial condition is concentration in the liquid phase. Does hydrus then create mass onto the sorbed phase to be in equilibrium with the liquid phase concentration via Kd, or does it pull mass out of the liquid phase to occupy the sorption sites?
Any help would be appreciated. Thanks.

Marek, 09/23/2006
I have always assumed a priori that it is the first case (other models do the same). You see, initial conditions must mean that the solution is at equilibrium with the solid phase. These conditions might be altered due to processes that you attempt to simulate, e.g. injecting water with the given concentration of the same solute. While this injected water mixes with the residing (initial conditions) water, new equilibrium conditions will be created that will be controlled by Kd.
Cheers


Jirka, 09/25/2006
Darren,
It is neither of those two options. While it would be easy to let the program calculate the nonequilibrium phase concentration (concentration in the immobile phase or kinetically sorbed concentration) to be in equilibrium with the liquid phase concentration, this would limit the use of the program. We actually allow users to specify as the initial conditions both the liquid phase concentration, as well as the concentration in the nonequilibrium phase (i.e., concentration in the immobile phase or kinetically sorbed concentration).
Jirka


## ID = 684, Constant Flux Boundary

m_north, 09/27/2006
I am simulating the wetted margin adjacent to an Antarctic lake. The goal of the project is to gain a better estimation of evaporation and flux of water through the active layer (above the permafrost). Using an atmospheric boundary for the top boundary condition does not give soil moisture values close to the measured. I have decided to use a constant flux boundary so that I can encourage differential soil moisture distribution across the wetted margin (the soil getting drier further from the water).

I recognize that the constant flux is usually meant for something like a rainfall rate (cm/d). However, I wonder is it also good as a net flux such as precipitation minus evaporation? In my case, evaporation is MUCH larger than precipitation with precip being >5cm/y. Thus, I have been inputting negative flux values. I want to make sure this is correct. Does the model see the negative flux values as a net evaporation?

Additionally, I have wondered why the velocity vectors have varying sizes since the color spectrum is the indicator of the magnitude. What then do the varying sizes represent?
Thank you very much for your assistance.
Melissa


Jirka, 09/28/2006
Mellisa,
What you do is correct, but I would not recommend that. When you use atmospheric BC, then the code takes the potential rainfall, substracts the potential evaporation and

applies the result at the boundary. Thus the resulting flux should be the same as your constant flux. However, the constant flux BC can not handle system dependent events (it is not intended to), such as ponding, or evaporation in excess of what the soil profile can deliver. Otherwise the results should be the same as for Atmospheric BC.

The length of velocity vectors reflects, similarly as the color spectrum, the magnitude of fluxes. You can manipulate that using Options->Velocity vectors parameters.
Jirka

02/07/2008 : 03:22:08
Hi There

In my previous 2-D simulations of septic tank project, I used time-variable top BC for the areas without septic tanks and constant flux top BC for the locations with septic tanks. The time-variable top BC has a zero solute concentration input, while the constant flux top BC has constant solute concentration input. The time-variable top BC is net soil drainage obtained from Hydrus-1D simulations (i.e. only rGWL column has values, zero values in Prec, Evap and Transp columns).

I now wish to see if I can save some running time by changing time-variable top BC to another constant flux top BC but with zero solute input. What is the best approach to do this?

Should I use different solute BCs to differentiate these two constant flux top BCs by setting "type 1" for the constant flux with solute input and "type -3" for the constant flux without solute input?

Or should I first simulate water flow only, using constant flux top BC (i.e. net soil drainage) and then import the final steady state pressure as initial condition for simulating solute transport?

Thanks for your help.

Liping

02/12/2008 : 18:01:35
Liping,

Well, you can assign different constant water fluxes on different parts of the boundary. You can also use on all these different parts the third-type (concentration flux) BC. You can differentiate between concentration values using different pointers and different positions in the cBound vector (cBnd1, cBnd2, etc). However, that (if you use either constant of time-variable flux with same flux values) will not make any difference in the computational time.

Finding first steady state solution and using it as the initial condition will save some computational time (as you will not have to be running water flow anymore), but the saving will likely be only about half of the computational time as solute transport stability criteria limit time steps for solute transport calculations. But this will depend on the actual conditions.

Jirka

**ID = 686, How to couple Hydrus 1D into Modflow**

Newman, 10/02/2006
I want to know how to couple hydrus1d into MODFLOW to simulate water flow and solute transport. Who can give me some information about it? Thanks very much.

Jirka, 10/02/2006
We are working with Sophia Seo and Eileen Potter (international Ground Water Modeling Center) on coupling water flow part of HYDRUS to MODFLOW. The HYDRUS package is almost finished, so is the documentation of it. It should be released to the public in a short while.

Navin Twarakavi (my postdoc from India) is in the meantime expanding this package to include solute transport for a possible further use with RT3D. I believe that we will be presenting it at the AGU meeting this fall in San Francisco.
Jirka

**ID = 687, How to download HP1?**

Newman, 10/02/2006
I am very interested in HP1,
how can I download it?

Diederik, 10/02/2006
Just go to:
www.sckcen.be\hp1
go to download, and fill in the form,
greetings,
diederik

**ID = 689, Cl influx from base of profile**

Kelley B., 10/03/2006 : 02:28:07
Hello,
I am using Hydrus to simulate increased recharge related to land use change. I am looking at wetting fronts versus Cl solute fronts for simulations using sand and silt with profiles of various depths and the water table at the base of each. I am using measured pressure and Cl concentration data for IC and running the simulations for 100 years.

Upper boundary conditions – solute, concentration, 0.006 mg/cm3; constant flux of 25mm/yr.

Lower boundary conditions – solute, zero gradient b/c it seemed to be the only one that would make sense for this situation (the others being a concentration, or concentration flux). Constant pressure boundary condition at the base, set at 0 pressure to represent the water table.

For most of the simulation, as the model works towards equilibration, there is a Cl influx from the base of the profile (as water moves upwards). The Cl concentration entering from the base seems to vary depending on the case, and even between timesteps.

Does anyone know if there is a way to control the Cl concentration moving up from the base?
Thanks!
(I have also set up simulations using a free drainage boundary condition – which eliminates the problem of increased Cl into the profile, because it drains - but I would like to know if I can set up a simulation with a constant water table...)

Jirka, 10/03/2006
Kelly,
If you have zero gradient lower boundary condition for solute transport, then there should no inflow of solute at the bottom. However, this boundary seems inapropriate for the upward flow (which I assume you are simulating). For Upward Flow you should specify the solute flux BC and provide the concentration of the Ground Water (which then multiplies the water flux to get the solute mass flowing into the profile). Zero gradient BC is good only for downward flux, but then there should not be any inflow.

Jirka

Kelley B., 10/03/2006
Hi Jirka,
Thanks for your response. The simulations work well using the concentration flux lower boundary condition. This allows me to control the concentration of Cl coming in through the bottom of the profile until the wetting front from the top and bottom meet and there is a downward flux at the base. Am I correct in thinking that the concentration flux lower boundary condition only has an effect if there is water moving into the profile? It shouldn't affect concentrations coming out of the profile, should it?
Thanks again,
Kelley
Jirka, 10/04/2006
Kelly,
If you specify concentration flux BC at the bottom of the profile with a given concentration, then this concentration is applied in both directions. The code takes

flux (in either direction) and multiplies it with the specified boundary concentration, resulting in the boundary concentration flux. That may not be appropriate for your problem when flux direction alternate, since for the downward flux this does not make much sense.

If you have a groundwater with some concentration (and flux alternating in both direction), you need to use the concentration BC at the bottom, when you specify directly the value of concentration on the boundary. Then if water flows upward water will have this concentration, if the flow is downward, then the code takes into account both GW concentration and concentration of water in the profile to give the actual concentration flux.

You may think that it would be logical to have concentration flux BC when the water flux is upward (good), and zero gradient when the water flux is downward. But I'm not so sure about this. First the code does not have such option although it would not be difficult to implement that. But with the zero gradient BC the code could not take into account the concentration in the GW, and it would seem as if the incoming water was instantaneously diluting GW (which is unlikely - larger body of water, larger lateral GW fluxes, etc, it is likely that GW concentration will be more or less unchanged by relatively small fluxes from the vadose zone). Any comments on this?

Jirka


Kelley B.,  10/09/2006
That makes sense Jirka.
Thank you so much for all your help.
Kelley


## ID = 690, Terminating not converging Hydrus runs

Besel   10/04/2006
Dear Hydrus user,
I am running Monte Carlo simulations with Hydrus1D using randomly sampled soil parameters. I am just wondering whether there is any option to terminate model runs, when the numerical solution is not converging (because unsuited parameter combinations are used). This would reduce the computing time of MC simulations considerably since these model runs have large water balance errors anyhow.
Thanks,
Benny


Jirka,   10/04/2006
Benny,
I'm sending you the version of the code that automatically stops and closes all the output files when it does not converge in the solution of the Richards equation

(instead of trying to continue). Let me know if you would need the code to do something else in such case, such as sending some message, ...
Jirka

**ID = 619, Units for the constant flux**

Vivekgalla, 10/06/2006 : 22:46:00
Helo all,
i would like to know the units for the Constant flux boundary condition in Hydrus-1D.And how do i specify the geometry of my soil profile in terms of width, i know there is one which asks for depth of the soil profile.
thanks
vivek

ChrisM, 10/14/2006
I think you cannot specifiy the width of your profile. You can decide whether your fluxes are vertically oriented or along an x-axis, but as far as I know, you can create a 1-dimensional model of the fluxes only, so HYDRUS will give you the fluxes at this point, not in an area.
Not that I am a specialist... probably Jirka has a better answer than me...

Regards,
Chris

Jirka10/14/2006
HYDRUS-1D is a one-dimensional program. Therefore its geometry is given only by the depth of the profile (in different directions, i.e., vertical or horizontal), and fluxes are in Lenght per time, e.g., cm/d.
J.

Vivekgalla, 10/23/2006
thanks jirka and chris, just got confused with the units.

**ID = 692, Convergence problem**

Tim, 10/12/2006 : 03:08:51
I'm using the Beta0.6 version to model water movement from an infiltration basin, through 115 meters of unsaturated sand, to the water table. Axisymmetric, rectangular grid, node spacing about 2.5 m, constant-head BC on the lower 10 m on the right (equilibrium) and 1 m constant head on the nodes representing the basin (25 m wide at upper left). Soil type is sand; I've tried the Sand, Sandy Loam, and Loamy Sand from the catalog.

Every time I try to run this, it starts out fine and runs for several days, then at random time steps it has trouble converging. I've increased the maximum number of iterations

to 35, so I can get it to run until the wetting front reaches the water table and builds up a mound, at the cost of long computation times. Even at that, the model gets stuck at about 27 days and cannot coverge.

The graph of iterations vs. time confirms that most steps are taking 3 to 5 iterations, then at random intervals one step will take more than 20. I don't understand why this would happen, since there are no atmospheric boundaries and nothing different is happening from one time step to another - it is just moving water through the soil at the same rate.

I also notice that the calculations take longer for each time step, even though the number of iterations stays about the same, after about 13 days. This is about when the convergence problems begin.

Time steps are in days, minimum step is 1e-6, water content tolerance 0.002 and pressure head tolerance 0.01, time step controls are model default values.

This is frustrating as it seems a simple setup and I don't understand why it should have difficulty at some time steps but not others.


Jirka, 10/12/2006
I'm sure that the problem is in the spatial discretization. If you have a ponded infiltration, then the infiltration zone for sands can be very sharp and water contents can change from the initial values to the full saturation within few cm or tens of cm. You obviously can not solve such problem on a discretization that is in meters (2.5 m). Thus to be able to solve this problem, you have several options:

a) use much finer discretization in the zone where water contents are changing from initial value to some value corresponding to saturation, or
b) adjust already the initial conditions to your boundary conditions, i.e., specify initial conditions at a slope from right to left (and with 10 m on the right and 1 m on the left)
c) try to relax the pressure head tolerance to about 0.05 cm.

I hope that this will help you.
Jirka


Tim, 10/12/2006
a) I just redrew the mesh with 3 times as many nodes in each direction, so node spacing is now less than 1 m, and it is running extremely slowly, about 5 minutes per day. At that rate it will take all day to get one simulation!

b) That won't accomplish the goal of this model, which is specifically to identify how long it will take for the initial application of water to pass through 115 m of dry sand and reach the water table. I set initial conditions to -5 m at the top and 0 at the water table with linear interpolation.

c) I tried that, still got problems.

I still don't understand why the convergence problems only occur at random intervals, beginning many days (typically 13 to 16) into the simulation. Seems like the problem would be worst at the beginning, when the gradients are highest, and get easier at later times - but the opposite is happening.

Jirka, 10/12/2006
I guess the convergence problems occur when mositure front reaches the next FE nodal level.
The pressure head criteria should be between 1-5 cm. I made a mistake in units in my previous posting.
J.

Tim, 10/22/2006
Still having trouble. I finally got the simple simulation to run by decreasing node spacing to 0.5m, and running the model overnight. Then I set up another domain to model the effect of anisotropy. This is a 125x125 m axisymmetric domain, all sandy loam. The upper 75m is isotropic. In the lower 50m, I set the second component of anisotropy to 0.1 so the Kx:Ky ratio is 10:1. BC as before, a constant head at the lower right corner and a constant head at the top left (the infiltration basin).

This model actually solved with nodes spaced at 2.5m and produced useful results. However, now I cannot get it to solve again, even though I have not changed any input paramters! It will run for maybe 20 days and then fail to converge.

One very odd thing: At the top boundary, the leftmost 10 nodes had -1m dz values (the bottom of the pond) and the rest are all at 0. When I view the results, on day 0 it looks fine, then as soon as infiltration begins, the top row of elements disappears from the 13th node all the way to the right. Now the top boundary looks like a dike between a shallow pond (on the left) and a slightly deeper basin to the right. It did not do this the first time I solved this model.

Did something change in the new release? I think it was the Beta 0.5 release that solved the model, but it will not solve now that I am using Beta 0.6.

Tim, 10/22/2006
Addendum to above: Hydrus does not appear to be correctly handling the condition when the maximum number of iterations is reached. According to the manual, it should reduce the time step interval and try again. Since I have the minimum time step set to 1e-6 (days), that should eventually get down to a small enough interval to achieve convergence. But I find that once it reaches that maximum number - no matter what I set it to - it will not converge on any successive try. Eventually it starts producing a message saying something about ORTHOMIN and reducing the time step; then it appears to converge on the next try, but immediately gives the same ORTHOMIN error message. Eventually either I terminate with Ctrl-C or, if I leave it running unattended, it produces gibberish.

Jirka, 10/22/2006

I'm sure that your problems stem from insufficient spatial discretization of the problem. Since I have written about spatial discretization so many times at the pages, as well as in the technical manual, I will not repeat it here again. I'm also pretty sure that the code handles well conditions when the maximum number of iteration is reached. It than divides the time step by 3 and tries again (until it reaches the minimum allowed time step). There is, however, no guarantee that the solution will be found with this smaller time step if the problem is not correctly defined or if the spatial discretization is not sufficient. ORTHOMIN is an iterative solver for the system of linear equations that is obtained after the discretization of governing equations. Errors in ORTHOMIN usually mean that the system of equation does not have a solution, which is usually an indication of the physical problem not being well defined or insufficient spatial discretization.

Jirka

Tim, 10/22/2006

I downloaded and installed the Beta07 version, and it solved the model that originally solved with Beta05 but would not solve with Beta06. This is a domain 125m square, with isotropic sandy loam in the upper 75m and 10:1 anisotropy in the lower 50m. Node spacing is 2.5m and, by increasing the allowable number of iterations to 40 and reducing the convergence criteria, it will run all the way to 350 days.

Now I am trying to solve the same domain, same criteria, same initial and boundary conditions, but with 3:2 anisotropy in the upper 75m. I reduced the node spacing to 1m - fails to converge after 22 days (after running for almost an hour). If I reduce the node spacing any further, the solution times become excessive. If I could change the node spacing only in certain areas, perhaps I could keep the total number of elements manageable.

I wish there was a way to see what exactly was going on with the iterations. VADOSE/W provides graphs during and after the solution, so you can diagnose problems more easily. Something like that might help.

It's still mysterious to me why the solution should only have convergence problems at random intervals, and why it has greater difficulty after several days, when nothing new is happening. Usually the big convergence problems happen early on, when the rates of change are greatest. HYDRUS is handling the first 10 to 20 days just fine, even though I have a constant pressure head of +1m adjacent to a region with -5m to -4m pressure head.

Tim, 10/25/2006

Here's another odd thing I just noticed. In the Calculation window, HYDRUS displays ItW (iterations for the time step) and ItCum (cumulative iterations). For a while these two numbers are consistent; ItCum increments according to ItW. But then there will be a step where the calculation takes a long time, and ItW will be a small number but ItCum will increment by a large number. For example, in the run currently calculating, in the 27th line ItW = 2 and ItCum = 97; the next line shows ItW = 5 and ItCum = 182. This happens repeatedly but apparently at random.

Jirka,   10/25/2006
This means that the number of iteration in a given time step was reached, the time step got decreased, and the solution was attempted again.
ItCum=ICum(previous)+n*IterMax+Iter,
where Iter is the number of iterations, IterMax is the maximum allowed number of iteration, n is "how many times the time step was decreased".

Tim,   10/25/2006
quote:
This means that the number of iteration in a given time step was reached, the time step got decreased, and the solution was attempted again.
ItCum=ICum(previous)+n*IterMax+Iter,
where Iter is the number of iterations, IterMax is the maximum allowed number of iteration, n is "how many times the time step was decreased".

Aha! So ItW only displays the number of iterations on the last (successful) try? That explains something I observed in an earlier message: when the specified IterMax appears in the ItW column, it means a solution will not be achieved. Seems like the calculation routine should stop and display an error message at that point.

Jirka,   10/25/2006
The code uses Picard implicit linearization method to find the solution. If even at the minimum allowed time step the solution is not found using the max allowed number of iteration, at the end the code will take one implicit step and continues with the new time step (again with Picard implicit method).
J.

Tim,   10/25/2006
Whenever the specified IterMax appears in ItW (e.g. if I specify a maximum number of 25, ItW will display 26), it is fatal. The program will continue trying with the implicit time steps for a while, then the ORTHOMIN error message will begin to appear. It has never occurred that the IterMax will appear and then the program will continue on to a solution.

I am beginning to realize that my problem is too big to solve on a PC. This machine has a 3.2GHz processor and 1 MB RAM, but when I reduce node spacing to 1 m it takes about half an hour to solve one day, because there are so many elements - and it still won't converge. To get a solution I would have to go down to 0.5 m or less, which would result in several hundred thousand elements. I need to model a time period of about two years, which would probably require more than a week of computation time on this PC.

Jirka, 10/25/2006

a) I would recommend to use first HYDRUS-1D to get a quick idea what discretization you need for your problem.
b) SInce flow in the vadose zone is mainly vertical, you need fine discretization in the vertical direction. However, the discretization in the horizontal direction may be much coarser. Perhaps you can try to make finer discretization in the vertical direction and coarser in the horizontal direction.
J.


Tim    10/26/2006
quote:
a) I would recommend to use first HYDRUS-1D to get a quick idea what discretization you need for your problem.

Yes, I often do that. In this case I have used HYDRUS to solve the simpler form of the problem, a homogeneous 50x125m column of sandy loam with 3:2 anistropy, with a node spacing of 1 meter. Now I am trying to solve the problem of interest, which involves a material change at a depth of 75m, using the same 1m node spacing in a 400x125m domain. The calculations become unstable and fail to converge when the wetting front is only 20 - 30 meters deep, long before reaching the material change.

quote:
b) SInce flow in the vadose zone is mainly vertical, you need fine discretization in the vertical direction. However, the discretization in the horizontal direction may be much coarser. Perhaps you can try to make finer discretization in the vertical direction and coarser in the horizontal direction.

Yes, I have been trying that too, but so far unsuccessfully.
In the problem of interest there is a silty layer at 75 - 80m, so there will be a perched layer and significant lateral movement. I was able to simulate this approximately by setting a higher anisotropy (10:1) in the zone below 75m, and that model ran succesfully with a node spacing of 2.5m (note that it would not solve with the Beta06 release, only with the Beta05 and Beta07 versions).
I have also had good results with a homogeneous model with 10:1 anisotropy throughout, using 2.5m node spacing. That solved fairly quickly.
Now my client wishes to know what the effect of the silty layer will be. We know that nonhomogeneous flow is different from homogeneous anisotropic flow, so we are interested to see how much difference it will make. So I am trying to include a discontinuous 5m thick sandy silt, expecting to see perched water and a spill-over effect. But so far I cannot get the recharge water even to reach that far down before the equations become unstable. With such a large domain, 1m node spacing generates 97 000 elements, and it takes several hours to find out that the solution will not converge... very frustrating.


Tim, 11/04/2006
Now this is interesting: If I select the Kosugi lognormal soil-hydraulic model instead of the Van Genuchten-Mualem, it solves just fine. Calculation times are drastically reduced, and there are no convergence problems. I'm getting some really interesting results now.

**ID = 693, Flow animation output?**

Tim,10/12/2006
I would love to be able to send my clients a QuickTime or Windows Media file, or even animated GIF, showing the Flow Animation from a Hydrus 2D run. Is there an easy way to do that?


Mirek, 10/12/2006
Currently there is no direct way how to export HYDRUS flow animation to *.avi or similar video formats. We are going to add this new option in version 1.01 that should be released at the end of November 2006. If you need to create the animation now then there are two ways:

a/ You need to use a program that can create an animation from single frames (for example the Fireworks from Macromedia or other programs). Start this program and at each time step of the flow animation in HYDRUS press Alt+"Print Screen". This will copy the HYDRUS window bitmap into the clipboard. Then paste (Ctrl+V) this bitmap into the program. Finally create the animation and export it to a file.
b/ Your customer could install HYDRUS demo version and use it as a "viewer". If you give him a project with results then he will be able to run the flow animation in his HYDRUS.
Mirek


Tim, 10/12/2006
Thanks Mirek, I'll try those.


Jirka, 10/12/2006
I got this advise from Naftali Lazarovic:
To make animated gif from HYDRUS results:
1. Open the graphical display of results in HYDRUS

2. Get a better view of your domain with view all button

3. Press on the Print Screen key (on your keyboard, usually it is above the insert key). By doing this the screen will be saved in the cash memory.

4. Open an image analysis software (like PHOTO EDITOR) and paste the cash as a new image.

5. Now you can copy only the domain and paste it again as a new picture. You can manipulate the background if you wish.

6. Save the picture in JPEG format

7. Repeat 1-6 for the rest of the print times ("…Everything should be as simple as possible, but no simpler…")

8. Open "Animator nine 3.6" or another software that can make animated gif (you can find this free software at: http://dw.com.com/redir?pid=909432&merid=61865&mfgid=61865<ype=dl_dlnow& lop=link&edId=3&siteId=4&oId=3040-2186_4-909432&ontId=2186_4&destUrl=http://www.download.com%2F3001-2186_4-909432.html

9. Press stitch and follow the instructions of the software

10. After you finish and close the "stitch" window you can press "Make" to save your animated gif file

You can also make presentations in PowerPoint using steps 1-6.


 s.zanarello, 10/18/2006
I suggest you the freeware CAM STUDIO 2.0. It's a screen registration software; it registeres your screen (whole screen or a windows) and exportes it in a video file.
You could use it to register the windows during HYDRUS animation.
The ling for the download is:
http://www.camstudio.org/


Jirka, 10/18/2006
Within a week we will post a version of HYDRUS (2D/3D) that will have an option to save the animation as a video file. Mirek has already implemented it and we are now testing it.
J.

Here is the text that we wrote for the online help:
Users can save the flow animation using the Create Video File command (Tools->Create Video File). This command calls the Create Video File dialog window, in which a user needs to specify where the Video File should be saved and under what name, type of the video file [a) Cinepac Codec by Radius, b) Microsoft Video 1, c) Intel Indeo Video 4.5, d) Intel Indeo Video 5.10, e) Microsoft MPEG-4 Video Codec V1, and f) Microsoft MPEG-4 Video Codec V2] and its Quality, and finally whether recording is carried out at Real Time (i.e., video will run at the same speed as HYDRUS animation) or only when frames in the View Window change (only changes in View Window are recorded). Additional options (such as Smoothness, Data Rate (i.e., kilobits per second)) are available for each particular video format.


Tim10/18/2006
quote:

b/ Your customer could install HYDRUS demo version and use it as a "viewer". If you give him a project with results then he will be able to run the flow animation in his HYDRUS.

We're having trouble with this. I sent him two projects that open just fine in my program, but when he opens them, there are no results to display. I sent him the .h3d file that opens, with results, in my program. Is there some other file I need to send?

Jirka10/18/2006

Tim,

In the "Project Information" dialog window, you have an option to have either a) Temporary or b) Permanent Working Directory. When the Working Directory is Temporary, then both input and output files are stored in the Project_name.h3d file. If the Working Directory is Permanent, then output files are stored only in this directory and not in the Project_name.h3d file. Thus if you want to send the project to your customers, you need to use the Temporary Working Directory option or send this directory along with the h3d file. See the description in the User Manual on pages 19-21.

Jirka

Tim, 10/19/2006

Okaaay... I read that section of the manual, and it says - several times - that all input and output data are stored in the .h3d file. Nowhere does it say what you just told me. I've read this section several times now and I still don't understand the purpose of the two options.

So what exactly do I send to my client? The entire directory with the input and output files, AND the .h3d file? And what exactly does he do (which files does he open) in I would just change the settings back to Temporary, but then wouldn't I have to re-solve? This particular model took hours to run. I'm not eager to do that again.

Mirek10/19/2006

Hello Tim,

here is the text from manual :

...

The Working Directory is a folder into which the program stores temporary data. Each open project has its own Working Directory, where the program stores, for example, input files for computational modules, and where computational modules write the output files. When saving a project, data from the Working Directory are copied into the main project file project_name.h3d. When the project is closed, the Working Directory is deleted. Only when a user selects the option "Permanent – result files are kept in this directory" (Fig. 4) is the Working Directory not deleted after closing the project, in which case the temporary data are not copied into the main project file.

...

So if you have results in the permanent directory then you should send both "Project_Name.h3d" file and "Project_Name" directory.

You can also change the Working Directory settings from "Permanent" to "Temporary". Then you should press "Save" and HYDRUS will copy all result files from the permanent directory to "Project_Name.h3d". However, I'd recommend you to backup your projects before you manipulate with them. Making backups regularly is very useful not only when working with HYDRUS but I think that this everybody knows very well...

Mirek

Tim, 10/19/2006
That doesn't answer either of my questions.
1. If I send my client the.h3d file and the Working Directory contents, how does he go about viewing the results? I guess he will need to copy the files that I send him into a directory and then use the Project Manager to tell HYDRUS where to find the files.
2. As I said, I have read that section of the manual, so quoting it back to me isn't helpful. I understand _what_ the two options do, but not _why_. Put another way, how does the user decide which option to select for a particular project? What advantage is there to saving all the information in separate files in a directory (Permanent option), rather than saving it all in one file (Temporary option)?

Jirka, 10/19/2006
By default, HYDRUS uses the "temporary" option. Then all the information is in the Project_name.h3d file and you can send this file to your clients to look at.
"Permanent" option is for users who actually do want to look at the input and output files directly and who want to do something with them. They can edit input files (ASCII) and import it back to HYDRUS, or they can use their own graphical editors to create graphs. Then it make sense to keep these input and output files in the "Permanent" folder. If this option is used, then the working directory must be copied together with the project_name.h3d file.

WOuld that be clear if I reformulate the text in the manual to:
Projects can be copied with the Project Manager only within a particular Project Group. Users can copy projects between Project Groups (or share their HYDRUS projects with colleagues and clients) using standard file managing software, such as Windows Explorer. In that case one must copy only the project_name.h3d file (when the radio buttons Temporary – is deleted after closing the project is used, Fig. 4). When temporary data are kept permanently in the working directory (i.e., the radio button Permanent –results files are kept in this directory is selected, Fig. 4), the working directory must be copied together with the project_name.h3d file.

Jirka

Tim,10/19/2006
quote:

By default, HYDRUS uses the "temporary" option. Then all the information is in the Project_name.h3d file and you can send this file to your clients to look at.

"Permanent" option is for users who actually do want to look at the input and output files directly and who want to do something with them. They can edit input files (ASCII) and import it back to HYDRUS, or they can use their own graphical editors to create graphs. Then it make sense to keep these input and output files in the "Permanent" folder. If this option is used, then the working directory must be copied together with the project_name.h3d file.

All of that (very helpful!) information should be added to the Help window for that dialog. (Currently that page says project_name.h3d contains all input and output data, which is not true if the Permanent option is selected.)

quote:
WOuld that be clear if I reformulate the text in the manual to:
Projects can be copied with the Project Manager only within a particular Project Group. Users can copy projects between Project Groups (or share their HYDRUS projects with colleagues and clients) using standard file managing software, such as Windows Explorer.

OK so far. I would rewrite the rest as follows:
If the "Temporary" working directory option is selected (Fig. 4), all input and output information is copied to the project_name.h3d file, and the user only needs to copy this file. If the "Permanent" working directory option is selected, then the input and output files are maintained separately and the project_name.h3d file will not contain all the information necessary to view the results. The user will need to copy the project_name.h3d file AND the contents of the Working Directory in order to save all of the input and output data.

To view the results of a solved project on a different computer, copy the .h3d file (and the Working directory files, if the Permanent working directory option is selected) into a Project Group directory (created with the File>Project Manager>Project Groups tab, Fig. 2). Use the File>Open command and browse to the Project Group directory, select the .h3d file of interest, and click Open.


Tim10/30/2006
Jirka,
The video output seems to work fine. Thanks for adding that.
Where can I find basic information about the different video file type options?


Jirka10/31/2006
I will let Mirek answer this question. He implemented this option int HYDRUS.
Jirka

**ID = 694, Advection vs transport calcualation duration**

Jkrivic, 10/18/2006

Hi all,

I'm simulating radionuclide migration from repository in phreatic zone.

The tunnel-type repository is situated in loamy sediments (K=1e-7 m/s). The K of concrete is approximated at 1e-9 m/s. The hydraulic gradient si very small, leading to Darcian velocities of 1e-9 m/d. In this case the transport calculations are very quick, with negligible mass-balance error.

When I try to simulate cases with increased K of the concrete (up to 1e-7 m/s), which leads to groundwater velocity of 1e-5 m/d, the calculation time increases so dramatically that any long-term calculation becomes impossible.

My conclusion is that diffusion dominated transport is much quicker to calculate than advection dominated transport for the same time frame.

Is there a way to circumvent the influence of increased advection on calculation time?

Jure


Jirka10/18/2006

Jure,

Please see section on "Oscillatory Behavior" in the manual that deals with this issue. Basically, the numerical solution may become unstable for problems when advection dominates over diffusion. To overcome these unstabilities, we need to start using smaller time steps. Check out the "Run Time information". If Courant number is equal to one (this links temporal and spatial discretization; small elements allow only small time steps), then the code cannot use larger time steps for the special discretization you are using. You would need to use coarser spatial distribution that would allow larger time steps. If, however, Courant number is smaller than one, the time step is likely govern by the performance index (or "Stability Criterion" in GUI, ie., Product of Courant and Peclet number). In that case you may try to relax this to, let's say, 10. Check the results and if they are still stable increase that further.

Jirka


**ID = 695, Capillary rise stimulation**

Oranse,10/24/2006

Hello, I am not familiar with Hydrus so I wanted to know if it can help me stimulate the following situation:

I take very dry sand (oven dry) and put it in contact with a water source at the bottom side. The upper side is open to the atmosphere.

Than I measure the weight gain vs. time.

I am using brooks and Corey function to describe the water retention curve.

Is it possible to stimulate this situation?


Jirka10/24/2006

No, HYDRUS can not "stimulate" this.

But Yes, it can "simulate" it.

J.

**ID = 696, Effective conductivity question**

Tleao, 10/26/2006
Hi,
I am performing numerical simulations with constant flux in various scenarios. However, at low water fluxes there seems to be a discrepancy between the calculated hydraulic conductivities from the numerical simulations data and the ones calculated from the pressure heads in the profiles using the Brooks and Corey model. Following is an example.

Experiment description:
Vertical plane 2D flow, clay-loam soil material, Brooks and Corey model
Profile Dimensions: 10 cm x 10 cm
100 h experiment
The flux through the profile is fixed at $10^{-9}$ cm/h
The initial heads are -100,000 cm for the upper boundary condition and -1,000,000 cm for the lower BC
After the software is run the fitted heads are -124,736.6 cm for the upper BC and -994,767.6 cm for the lower BC. From the fitted pressure heads we calculate the effective hydraulic conductivity as:

K = q/(dh/dL)
q = $10^{-9}$ cm/h
dh = -124,736.6 - (-994,767.6) = 870031 cm
dL = 10 cm
calculated K = $1.15 \times 10^{-14}$ cm/h

However, this conductivity is lower than the one calculated from the Brooks and Corey model using the lower pressure head in the profile, -994,767.6 cm, which is equal to $3.3 \times 10^{-13}$ cm/h.
Any thoughts?
Thanks,
Tairone

Jirka, 10/26/2006
You may well be below the water content tolerance. If you want to have very precise calculations, select very small water content tolerance and disable the interpolation tables (make both value equal to zero).
J.

Tleao, 10/31/2006
I changed the precision for water tolerance and disabled the interpolation tables, but it did not make much difference. It might be because I am working with simulations with pressure head as initial condition instead of water content. Is there a way of increasing the accuracy in the pressure heads, for very large values?
Thanks
Tairone

Jirka, 11/01/2006
Tairone,
The water content tolerance is used as a convergence criterion in the unsaturated zone, and the pressure head tolerance in the saturated zone. Thus increasing precision in the unsaturated zone can be done only by changing the water content tolerance.

Jirka

## ID = 697, How to convert flux to concentration

Ning, 11/02/2006
I use hydrus1D to calculate the chloride concentration that go through vadose zone to saturated zone. The result is the cumulative bottom solute flux, right? Its unit is $g/m^2$ but I want the unit that is mg/L. How to convert it?

Thank you very much
Ning

Jirka, 11/02/2006
Cumulative bottom solute flux, Qc, is the time integral of q*c (i.e., q*c*t). Thus you can get average concentrations by deviding Qc by cumulative water flux Q (q*t). Make sure that unit conversions are correct.

Jirka

## ID = 398, 3D geometry question: angles layers

Brian,11/02/2006
I have been utilizing the User Manual to attempt to create a 3D model with 2 layers, with the boundary between the 2 layers being angled from one side to the other. I have been struggling with understanding how do create such a model in HYDRUS. I have tried creating 2 layers in a solid with variable thicknesses, but cannot figure out how to actually vary the thicknesses of those 2 layers. Any help would be greatly appreciated.

I would like a step-by-step set of instructions as to how to create such a model, with the following geometric specifications:

Total length in x-direction: 74 m
Total length in y-direction: 42 m
Total length in z-direction: 14 m

Layer 1 (the lower layer) is bounded by the following (x,y,z) coordinates, in meters:

Top of Layer 1: (0,0,12.1), (0,42,12.1), (74,0,10.6), (74,42,10.6)

Bottom of Layer 1: (0,0,0), (0,42,0), (74,0,0), (74,42,0)

Layer 2 (the upper layer) is bounded by the following (x,y,z) coordinates, in meters:

Top of Layer 2: (0,0,14), (0,42,14), (74,0,14), (74,42,14)
Bottom of Layer 2: (0,0,12.1), (0,42,12.1), (74,0,10.6), (74,42,10.6)

Of course, the coordinates for the bottom of Layer 2 are the same as the coordinates for the top of Layer 1.

I am guessing that this is an easy thing to do, but I am having trouble understanding the User Manual instructions for this task.

Thanks.
Brian


Mirek, 11/03/2006
Hello Brian,
I'll create a short video tutorial. It will be available at http://www.pc-progress.cz/Fr_Hydrus3D_Tutorials.htm later today. I'll let you know.
Regards Mirek


Mirek11/03/2006
Hello Brian,
The video tutorial Solid_08 has been already published, you can find it at http://www.pc-progress.cz/Fr_Hydrus3D_Tutorials.htm.
We are currently working on documentation (the User Manual and Help) and we would like to complete it so that things like this were clearer (although I think that the best way how to explain some actions are the video tutorials). The completed documentation should be available in version 1.01 that will be released at the end of November.

The tutorial Solid_08 explains how to define Sub-Layers with variable thicknesses in projects which have just a few thickness vectors. However, this method would not very good in projects with thousands of thickness vectors - for example if the domain geometry is imported from a GIS program. Therefore we have added several new options that allow defining such domains efficiently. These options will be available also in the version 1.01.

Best regards
 Mirek


Brian11/03/2006
Good Day Mirek,

Thank you very much for the quick response. I will view the tutorial and then apply the method to my model. I agree that video tutorials can sometimes be the best way to communicate a multi-step process such as this.

Brian

## ID = 699, Kosugi more robust?

Tim11/04/2006
Has anyone else found HYDRUS to be more robust and faster to solve when the Kosugi lognormal soil-hydraulic model is selected instead of the Van Genuchten models? It seems to handle everything I throw at it - wide node spacing, sandy soil, high permeability contrast, rapid flux changes - where the VG or VG-Mualem options have major convergence problems. For setups that solve with the VG-M option, the Kosugi method leads to much faster solutions, but the results appear to be similar. Is this consistent with others' experience?

Jirka11/04/2006
I do not think that many people were using Kosugi's functions. I have actually not seen myself any papers where HYDRUS would be used together with these functions. I also usually use VG-M model. It is possible that for soil you have in your project, Kosugi's functions are less nonlinear (the conductivity function?) than VG-M model. Jirka

Tim11/05/2006
I'm using soil types in the HYDRUS catalog - sandy loam and loamy sand - for the most part. Applying Rosetta to textural and bulk density data for site soils produced values quite similar to the parameters for Sandy Loam in the HYDRUS catalog.

When I switch to the Kosugi method, Rosetta is turned off, but HYDRUS retains the parameters for the selected soil. I don't know if there is another parameter-estimation model that uses the Kosugi equations that I could apply to my site soil data.

I'll have to export the data for the K and Theta curves from HYDRUS using the VG-M and Kosugi models and see if they differ. Reading Kosugi's paper, I would not expect any significant difference in the curves. But I don't know how the Kosugi function is implemented in HYDRUS - if it substitutes a lognormal function for a nonlinear equation, that could be the reason it is more stable.

Since the Kosugi functions yield solutions in cases where the VG-M will not, it would appear to be a significant enhancement for HYDRUS. We should figure out how this works so that more people can make use of the Kosugi model if it is appropriate.

Thanks,
Tim
Jirka11/05/2006
Tim

When you switch the soil hydraulic properties model from the VG model to Kosugi, the GUI retains the physical parameters, i.e., the residual (Qr) and saturated (Qs) water contents, the saturated hydraulic conductivity Ks, and the tortuosity parameter l. It does not retain the alpha and n parameters (It just make alpha=1m and n=1). The user itself has to enter these two parameters for his particular soils. You could see that once you use the Kosugi model, the Neural Network Predictions (Rosetta), as well as the Catalog of soil hydraulic parameters (Carsel and Parish) disappear. This is because Rosetta works only for the VG model and the catalog is available only for VG and BC models. Nobody has so far did this work for the Kosugi model. I may try to do in the future the catalog for the Kosugi model, but at present such information does not exist.

Jirka

Jirka11/05/2006
Tim,
I took the soil hydraulic parameters from the catalog given in HYDRUS(Carsel and Parish parameters for the van Genuchten's model), generated retention curves (in the i_check.out file), and then fitted these retention curves using the RETC program and the Kosugi's model. I came out with the following shape parameters for different textural classes:

Soil Hydraulic parameters for the Kosugi's model:

Alpha (cm) n
Sand: 303.6932 0.3827
Loamy Sand: 12.4656 0.9497
Sandy Loam: 27.4229 1.2600
Loam: 101.7962 1.7997
Silt: 510.5564 2.4804
Silt Loam: 325.8513 2.3017
Sandy Clay Loam: 80.8916 2.0374
Clay Loam: 666.3264 2.8133
Silty Clay Loam: 2852.6181 3.2643
Sandy Clay: 1129.4786 3.4099
Silty Clay: 140538.00 4.4946
Clay: 103815.28 4.6689

I hope that this is useful.
Jirka

Tim11/06/2006
Thanks Jirka! I had missed the fact that the parameters were reset when I switched to the Kosugi model.

I used the parameters you provided for Loamy Sand and Sandy Loam and re-ran the model. Now the results are drastically different... and the curves generated by those

parameters don't look right for sand, they look more like clays. Are you sure these are right? I need to be sure about this.

Tim11/06/2006

Argh. I just realized the problem was units - you gave me Alpha in cm and my domain is in meters... So I corrected and re-ran, and now I am back to having convergence problems again! Apparently the Kosugi model doesnt work miracles after all. I'll spend a little more time with this and report back.

## ID = 700, Whats the meaning of "level" in SOLUTE.FOR?

Rayme, 11/05/2006

hi,

May I ask a question about the source code in "solute.for" (partly listed below): what's the meaning about the symbol

"level", such as "if(Level.eq.1) then ... else ....end if"?

```
do 11 i=2,N-1
a1=b1
b1=x(i+1)-x(i)
dx=(x(i+1)-x(i-1))/2.
if(Level.eq.1) then
F(i)= Conc(jS,i-1)*......
else
B(i)=epsi*...
D(i)=dx/dt*thN(i)*Retard(i)+...
E(i)=epsi*...
F(i)=F(i)+...
end if
11 continue
```

I learned that the source code can be compiled, linked and run from 3.0 version of GUI of HYDRUS-1D. When I tried to do it I met some proble(listed below). How could I compile, build, and excute the program under the environment of the 3.0 version of GUI of HYDRUS-1D? Please give me a hand. Thanks.

```
--------------------Configuration: HYDRUS - Win32 Debug-------------------
Linking...
HYDRUS.OBJ : error LNK2001: unresolved external symbol _TMCONT@64
HYDRUS.OBJ : error LNK2001: unresolved external symbol _ALINF@28
HYDRUS.OBJ : error LNK2001: unresolved external symbol _OBSNOD@48
HYDRUS.OBJ : error LNK2001: unresolved external symbol _TLINF@240
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SOLUTE@268
HYDRUS.OBJ : error LNK2001: unresolved external symbol _TEMPER@104
HYDRUS.OBJ : error LNK2001: unresolved external symbol _WATFLOW@352
HYDRUS.OBJ : error LNK2001: unresolved external symbol _VELOC@52
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SUBREG@180
```

HYDRUS.OBJ : error LNK2001: unresolved external symbol _NODOUT@84
HYDRUS.OBJ : error LNK2001: unresolved external symbol _PROFIL@60
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SETSNK@128
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SINKIN@80
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SETBC@92
HYDRUS.OBJ : error LNK2001: unresolved external symbol _CHEMIN@112
HYDRUS.OBJ : error LNK2001: unresolved external symbol _TEMPIN@48
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SETRG@36
HYDRUS.OBJ : error LNK2001: unresolved external symbol _ROOTIN@28
HYDRUS.OBJ : error LNK2001: unresolved external symbol _TMIN@84
HYDRUS.OBJ : error LNK2001: unresolved external symbol _SETMAT@144
HYDRUS.OBJ : error LNK2001: unresolved external symbol _GENMAT@60
HYDRUS.OBJ : error LNK2001: unresolved external symbol _FH@12
HYDRUS.OBJ : error LNK2001: unresolved external symbol _MATIN@120
HYDRUS.OBJ : error LNK2001: unresolved external symbol _NODINF@120
HYDRUS.OBJ : error LNK2001: unresolved external symbol _BASINF@224
HYDRUS.exe : fatal error LNK1120: 25 unresolved externals
Error executing link.exe.
HYDRUS.exe - 26 error(s), 0 warning(s)

Jirka11/05/2006
When the governing convection-dispersion equation is discretized, the coefficient of the matrix are evaluated at two, previous and current, time levels. Thus Level=1 is the previous time level, and Level=2 is the current time level.
Jirka

## ID = 701, Rain input

Jick,11/07/2006
Hello everybody, I'm working with Hydrus-2D software for school. Basically, I have to simulate a drained field, with some other details.
My question is how to put rain data into my model. I would like to enter a few (rain)showers, but I have no idea how to enter it.
Anyone ;)?

Jirka, 11/07/2006 :
Try to use the atmospheric boundary condition.
J.
PS: Before asking questions in this forum, try to spend at least a little bit of time to understand the HYDRUS software. Look at the existing examples (that do have atmospheric BC in it; actually for Dutch conditions). We are not solving homeworks here.

## ID = 702, Heat transport

Sander Huisman, 11/07/2006
Dear all,
I am trying to understand the boundary conditions for heat transport. In particular, the cauchy boundary condition given in equation 4.10 of the manual. Is this equation OK? Do the left-hand and right-hand side q have the same meaning, or should they be q and q0 as with the cauchy boundary condition for solute transport?

To test the boundary condition, I simulated evaporation from an initially wet soil (no infiltration). This resulted in high surface temperatures. Can anybody explain this in words to me, but refering to equation 4.10.

I also made a model run simulating infiltration of hot water in a cold soil by specifying a high value for tTop. The soil temperature did not increase as I expected. Am i missing something here?

In general, I would also be interested to know when the Cauchy and Dirichlet boundary condition should be used for heat transport. For example, when only air temperature is available.

I guess what I really want to know is how good Hydrus approximates the fully coupled equations of Penning and de Vries. I am aware that Milly (1984) showed that ET simulations are quite reliable if all coupled terms are neglected. But what about surface temperature. And what about the specific implemtation in Hydrus (e.g. hCritA, boundary conditions). Does anybody have experience with this. Some old papers I might not be aware of?

Any help on any question will be appreciated.
Sander


Jirka, 11/07/2006
Yes, there should be a subscript "0" with the flux term on the right side of the equation 4.10.

Dirichlet BC means that you fix temperature in the boundary node.
Cauchy BC means that water infiltrating (or being extracted) has specified temperature. Thus if you specify infiltrating water of certain temperature, then this energy is used to heat up the soil (which may have a large heat capacity and thus the temperature increase does not have to be dramatic).

General version of HYDRUS does not consider all the coupling terms between the energy and water (vapor) flow. We do have a version that can do that, but that will be likely released in 2007. See the following article that describes this coupled version:

103. Saito, H., J., J. Šimùnek, and B. Mohanty, Numerical analyses of coupled water, vapor and heat transport in the vadose zone, Vadose Zone Journal, 5, 784–800, 2006.

Jirka
 Sander Huisman, 11/07/2006
Hi Jirka,

Thanks for the reference. I read it already. Interesting.
Could it be that something is wrong with the Cauchy boundary condition for heat flow?
Consider the following run:

Infiltration of 80 degree water (5 mm per day) in a loam soil with hini of -100000 mm and an initial temperature of 20 degrees and no evaporation. The soil profile is 1 meter long, discretized in 100 layer with a higher density of 0.01 at the surface. I simulated for a period of 100 days which wets up the entire profile. The lower boundary condition is fixed temperature at 20 degree.

It is my feeling that the soil should heat up considerably. However, the temperature varies only between 19.99 and 20.01. What do you think? Could it be that the default values provided by the interface for the heat capacity are wrong, they look awfully big with a Cw of 3.12e13 for the units mm and day. The chung and horton parameters look big too.

Sander


Jirka, 11/07/2006
Sander,
I have done exactly what you described above and got significant increases in temperatures throughout the profile. I will email you this project.
Jirka


Sander Huisman, 11/07/2006
Hi Jirka,
Thanks for your time.
This is going to sound silly... After I installed the latest version of Hydrus 1d on my computer all problems are solved. Even the old project that I set up here runs well with the latest version. I can check at my work which version was not working if you are interested, but perhaps it is not of interest.

Sander


Sander Huisman11/09/2006
Hi Jirka,
I ran the project you send me in the office with Hydrus version 3.00. It gave different results than the results you send me. Just for your information.
Sander


**ID = 703, Conc in post-processing basic profile information**

Anabdel,11/07/2006

Hello guys,

I have a question about the profile concentration of any solute you run transport simulation for. In the post-processing basic profile information (depth vs conc.) results, what does the program calculate the concentration for? Is it the concentrations in the soil (solid part) or in the solvent (liquid part/water) under flow or combined concentration?

Thanks.

Jirka11/07/2006

Program shows

a) dissolved concentration in the liquid phase, c, and

b) if there is nonequilibrium transport, then concentration in the nonequilibrium phase. This can be either dissolved concentration in the immobile water or the sorved concentration on kinetic sites.

J.

## ID = 704, Flow and transport of polluted rain

julie j,11/08/2006

Dear Jirka

I am trying to simulate pollutant flow and transport in the vadose zone. For this, I am working with a monthly rain, using the Time Variable Boundary Conditions Precip. and Evap in HYDRUS.

I have seen in the results of the simulation that the Actual Surface Flux is either positive or negative. After calculations I also observe that this variable is equal to Precip.-Evap.

Therefore I have several questions regarding the positive or negative flows:

- When the Actual Surface Flow is negative, does it mean that the water stored in the soil has evaporated ?
- And when the soil is dry, in summer for instance, how does the water evaporate ? and what does Prec.-Evap physically represent ?

I would also like to understand what happens in terms of transport.

-What does the pollutant contained in the rain become when Prec.<Evap. (negative actual flux)? Does it not infiltrate and therefore never enter the soil? Or will it be taken into account at the next time step when the actual surface flux will be positive?

- Does the evaporated water contain pollutant in the same concentration than in the soil? The concentration in the soil would therefore be unchanged?
- Is the water evaporated pure and the pollutant stored in the soil? The concentration in the soil would therefore increase?

I had otherwise thought of another way to solve the problem:

If Precip.<Evap, what do you think of considering that there is no actual surface flux and that the pollutant of the rain will be transported during the next time step when Precip.>Evap. ?

Thank you very much.
Julie


Jirka, 11/08/2006
Julie,
Water Flow:
Applied surface flux (i.e., potential surface flux) is equal to Evap-Prec. Actual Surface Flux is negative for infiltration and positive for evaporation.

When the soil is dry, then very likely the actual surface flux (evaporation) will be smaller than the potential surface flux (evaporation). How is the reduction calculated is written in the manual. The flux BC is changed to the head BC (with hCritA).

Solute Transport:
I use the following calculations to concentrate infiltrating water due to the effect of evaporation. As you can see, when Evaporation is smaller then precipitation, then the same mass of solute enters the soil profile (Prec*cT), and when Evaporation is larger than Precipitation no solute enters the soil profile. I do not consider this in the next time step.

if(Prec-Evap.gt.0.) then
cTop=Prec/(Prec-Evap)*cT
else
cTop=0.
end if

During evaporation, no solute leaves the soil profile. Solutes are left behind in the soil and concentrations increase.

If Precip.<Evap, what do you think of considering that there is no actual surface flux and that the pollutant of the rain will be transported during the next time step when Precip.>Evap. ?
This can obviously be done by users at the input.

Jirka


## ID = 705, Root water uptake problem

koenverbist11/08/2006
Dear Jirka,
Hydrus2D/3D apparently allows 2 alternatives to describe the root distribution, a first one by manually assigning nodes a relative value (0-1) and a second one that uses the root distribution functions of Vrught et al (2001).

When using the second option, I thought I should select the nodes to which I want to apply the root distribution function, give them a value 1 and then set the individual parameters of the root distribution functions. The interface, however, eliminates the 1-value from these nodes, when making adaptations in the root distribution function, and thus no root water uptake is generated.

When first setting the function parameters and then setting the nodes to which they should apply, no effect of the selected values is observed in the simulated actual root water uptake.

Could you please indicate how this second option should be used? Both the manual and the help function don't give much information about this feature.

Kind regards,
Koen.


Jirka11/09/2006
Koen,
The equations that are used to calculate the root distribution are given in both technical and user manuals, and in the online help. The equations are given and used in absolute coordinates, i.e., they are independent of any actual selection in GUI. The x and y coordinates are identical to x and y coordinates for the geometry of the transport domain. The only exception is that the beginning of the z coordinate for the root distribution starts at the highest located node in the transport domain (again independent of any actual selection). This function was developed for simple domain as given in papers of Vrugt or used in papers of Gardenas or Hansen.

Vrugt, J. A., J. W. Hopmans, and J. Šimùnek, Calibration of a two-dimensional root water uptake model, Soil Sci. Soc. Am. J., 65(4), 1027-1037, 2001.

Vrugt, J.A., M. T. van Wijk, J. W. Hopmans, and J. Šimùnek, One-, two-, and three-dimensional root water uptake functions for transient modeling, Water Resour. Res., 37(10), 2457-2470, 2001.

Gärdenäs, A., Hopmans, J. W., B. R. Hanson, and J. Šimùnek, Two-dimensional modeling of nitrate leaching for various fertigation scenarios under micro-irrigation, Agric. Water Management, 74, 219-242, 2005.

Hanson, B. R., J. Šimùnek, and J. W. Hopmans, Numerical modeling of urea-ammonium-nitrate fertigation under microirrigation, Agric. Water Management, (submitted December, 2005, revised April 2006, in press).

Jirka


koenverbist11/09/2006
Jirka,
I see, that explains why the second option doesn't work for me. I will stick to the manual assignment of the root distribution, then.

Thanks for helping me out,
Koen.


Onurakay, 02/20/2007
Hi. I have been simulating a soil column where a seed (treated with a slow-release pesticide) is planted and the growth of the plant and the solute concentrations are monitored over 150 days period. The top atmospheric boundary condition allow me to put potential transpiration rates. Since the plant is growing the root water uptake parameters change in time. I have the potential and actual water and solute uptakes from a previous RZWQM run. How can i incorporate that information into Hydrus without going into putting root parameters? thanks.

Onur


Jirka02/20/2007
HYDRUS does not allow you to change the root distribution. It considers that the root distribution is constant in time. It will allow you to specify time-variable potential transpiration and evaporation. The solute uptake will depend on the actual transpiration (calculated by the model) and the solute distribution in the root zone. You can not specify the solute uptake demand. I'm working on such option now (so that users can enter the nutrient demand, and that is supplied by both active and passive uptake). Currently the code considers only passive nutrient uptake.

Jirka


Onurakay, 02/20/2007
If we talk about just water flow, do you think it's possible if I can specify an internal sink term that will represent the actual transpiration rates over time?
Onur


Jirka, 02/20/2007
Onur,
You need to specify potential transpiration rate and spatial root distribution function. The potential transpiration will be fullfiled depending on water conditions in the root zone (stress response function of Feddes). You can adjust Feddes function so that your potential transpiration is equal to actual transpiration (as long as there is water available).



**ID = 706, Boundary condition**

Ning, 11/09/2006
Thank you Jirka for your answer. This is my first time that I do model, so it's my first time for Hydrus also. As I told you in last topic that I use hydrus for tsunami situation

to answer the questions that how much salt in terms of mass and concentration which go through to aquifer. My problem is constant BC (head and concentration) are fixing the boundary. Should we extend the boundary past the water table and use results from short distance above (using observation point) or would use zero gradient (free drainage bea better approach)?

Input Conditions:
Water flow boundary condition
- Upper boundary condition -> Atmospheric BC with surface layer
- Lower boundary condition -> Constant pressure head (zero)
- Initial condition -> In the pressure head
Soulte transport boundary condition
- Upper boundary condition -> Concentration BC (seawater concentration)
- Lower boundary condition -> Concentration BC (zero)

Another question : Cumulatvie botom flux
I think it is possible for the cumulative botom flux curve to level, then go down again with time based on flux rates during simulation, yes?
is there any situations when curve- will it go up (show decrease) , or change sign, or is that situation not possble given definition of cum bot flux?

Jirka11/09/2006
I do not really understand what your first question is. It seems that your input conditions are reasonable.

Second questions: Anything can happen under these boundary conditions. By fixing pressure head at the bottom, you do not control whether the flow is up (capillary rise) or down (recharge). This will depend on the atmospheric conditions (upper BC) and soil hydraulic properties.
Jirka

**ID = 707, Request code changed!**

newuserid11/13/2006
Hi, Jirka,
I got activation codes. After inputted the key1, I wrongly pressed the button "Activate Now" without inputting the key2! A information dialog appeared, and then the request codes changed! Now I can not activate the software anymore! How can I do? How to restore the previous request codes? If we reinstall OS on same machine, whether the request codes will also change?

Mirek, 11/13/2006
Hello,
1/ The request code1 changes always after two unsuccessful attempts to activate HYDRUS with incorrect codes (a warning message is displayed after the first failure).

2/ I don't know who you are and therefore I can not help you directly. Contact your reseller and ask him for new activation codes. However, your request code 2 should be same as before otherwise it means that new request codes were generated on a different computer.

3/ Under certain circumstances the authorization can remain valid even after reinstallation of OS but users should not rely on that. HYDRUS should be deactivated before any action that could cause problems with authorization – see "HYDRUS reinstallation or moving to another computer" at http://www.pc-progress.cz/Fr_Hydrus3D_Ordering.htm.

Mirek


newuserid, 1/15/2006
Hello, Mirek,
1. I'm not sure when the request code1 was changed. I entered Key1, and a warning message displayed. Then I entered Key1 and Key2 carefully, and a failure notice appeared.
2. I uninstalled and installed HYDRUS time after time, but the request codes were mutative. Unfortunatly, I find the request code2 are also changed today whereas I didn't change hardware in the same machine.
3. I tried to use HYDRUS with the 30-days license in notebook. How to install and use HYDRUS on two computers with the same license?


 Mirek 11/15/2006 :  10:59:11
My advice is very simple: if you have any problems with authorization contact your HYDRUS reseller and ask him for new activation codes.
I don't know who you are and I'm not willing to continue in this discussion on an anonymous level.

Mirek


### ID = 708, Evaporation and transpiration

Michaela,11/14/2006
I try to model soil water content for a period of more than 40 years. But I have some problem defining input data in the variable boundary condition dialogue box. What method to calculate potential evaporation do I have to use for generating input data? Is it realy potential evaporation what is ment or potential evapotranspiration instead?
thanks for help and best regards, michaela


Jirka11/14/2006
Michaela,
The potential transpiration is the atmospheric demand, which depends on meteorological variables, such as the net radiation, air temperature, wind speed, and relative humidity, and which does not consider actual conditions of the plant and soil.

The potential transpiration rate can be calculated from the meteorological variables using various process-based or empirical formulas, such as the FAO-recommended Penman-Montheith combination equation [FAO, 1990] or the Hargreaves formula [e.g., Jensen et al., 1997], respectively. As input to HYDRUS-1D, you need to split the potential evapotranspiration into potential transpiration and potential evaporation, e.g., using soil cover fraction or Leaf Area Index. The code then calculates actual transpiration and actual evaporation depending on soil conditions (saturation status). J.

Food And Agriculture Organization of the United Nations, Expert consultation on revision of FAO methodologies for crop water requirements, ANNEX V, FAO Penman-Monteith Formula, Rome Italy, 1990.

Jensen, D. T., G. H. Hargreaves, B. Temesgen, and R. G. Allen, Computation of Eto under nonideal conditions, J. of Irrig. Drainage, 123(5), 394-400, 1997.

s.zanarello11/20/2006
Hi Jirka
could you explain how to split evapotranspiration in transpiration and evaporation?
Is it a simple process like this reported below?
transpiration = evapotranspiration * Leaf Area Index
evaporation = evapotranspiration * (1-Leaf Area Index)

thanks for your help

Jirka, 11/20/2006
No, it is not that simple. You need to first calculate the "Surface cover Fraction" from LAI:

SCF=1.-exp(-rExtinct*LAI)
rExtinct=0.463

and then

transpiration = evapotranspiration * SCF
evaporation = evapotranspiration * (1-SCF)

Jirka

s.zanarello, 11/21/2006
Thank you Jirka. Effectively, it was too simple...
Where can I find deepenings about the correct approach? Is it in FAO publication?
If not, could you cite some literature reference where I can find this approach?
Thank you very much.

Stefano

Jirka11/21/2006
I believe that the formula is called the Beer's law, which partitions the solar radiation component of the energy budget via interception by the canopy. I also believe that it was referenced by Ritchie [1972].
Ritchie, J.T., Model for predicting evaporation from a row crop with incomplete cover, Water Resources Research, 8(5), 1204-1213, 1972.
J.


s.zanarello11/22/2006
thanks


uwollsch  12/18/2007
--------------------------------------------------------------------------------
quote:
Originally posted by Jirka

No, it is not that simple. You need to first calculate the "Surface cover Fraction" from LAI:
SCF=1.-exp(-rExtinct*LAI)
rExtinct=0.463
and then
transpiration = evapotranspiration * SCF
evaporation = evapotranspiration * (1-SCF)
--------------------------------------------------------------------------------

Hi Jirka,
could you please explain where the value of rExtinct=0.463 results from? Where can I read more about this?
Thank you,
Ute


 Jirka,  12/18/2007
I do not have this reference on hand right now, but I believe that this formula comes from:
Ritchie, J.T., Model for predicting evaporation from a row crop with incomplete cover, Water Resources Research, 8(5), 1204-1213, 1972.
Jirka



**ID = 709, Rate of internal nodal charge**

Srilert  11/15/2006
Dear all,
I set my 2 dimensional tank of soil and I injected tracer(NaBr) using peristatic pump with 12 ml./hr into soil. For this experiment, I simulate the injected tracer as a specify

recharge in the internal node in HYDRUS 2D. Then, I have to input rate in this node. I input rate = 12/((22*0.05^2)/(7*4)) = 6109 cm/hr(diameter of rubber tube = 0.05 cm) , is it correct ? I feel it 's very large value. How should I do ?
Thank you for your assistance.
Srilert


Jirka11/15/2006
Lert,
How long is your tank in the third dimension (perpendicular to the two-dimensional section)? You should just divide your flux (12 ml/hour) by the depth in the third direction and apply the resulting value.
Jirka


Srilert, 11/16/2006
Dear Jirka,
My tank have 8 cm. in third direction(80cm. long, 30 height, 8 cm thick ) . You mean the flux = 12 millilitres/hour diveded by 8 cm. equals 1.5 cm^2 / hour ?
I am wondering that my tube for release tracer having 0.05 cm. diameter (area = 0.001964 cm^2) why does flux equals 12 divided by tube area (0.001964 cm^2). I am sorry that I ask you again.
Best reagrds.
Srilert


Jirka11/16/2006
Lert,
If you indeed describe your internal source in the geometry of the transport domain, i.e., circle, then you do need to take into account its actual dimensions (radius, circumference) and apply the flux accros the length of the source (in L/T). If you, however, model that simply as a single internal node, then you just have to make sure that the right volume is applied (in L2/T).
Jirka


## ID = 710, Profile information

Michaela,11/16/2006
I have some problems with the Profile-module in Hydrus 1d. Unfortunately I do not understand how to edit my profile and reading the manual does not help me at all. Is there a more comprehensive user guide available on this topic?
best regards,
michaela
Jirka, 11/17/2006
Michaela,
Do our tutorials (posted somewhere on this site), especially the "Possible additional modifications" part (multiple materials, different discretization). That should get you started.
Jirka

**ID = 712, Editing time variable boundary conditions**

michaela11/21/2006
Running Hydrus 1d on a pc with german system software I have still some trouble editing the time variable boundary conditions. Even when I pre-define the period as decimal point, the VCI workbook designer changes the cell value into date format. The problem does not exist with data like 0.xx but with data like 2.1 or 14.5. It does not matter if I import data or type data into the cells dierectly.
Defining the perido as decimal point in the system software also did not solve the problem. Does anybody has some experience with this and can help me to solve the problem?

thanks michaela

Jirka11/21/2006
Michaela,
Please contact Martin Volkmann [volkmann@ifbk.uni-hannover.de] from the University of Hannover. I remember that while teaching there a HYDRUS course we had similar problem with the German version of Windows, but that we somehow resolved it. I just do not remember how.
J.

**ID = 713, Rectangular domain discretization**

msakai11/23/2006
Hello,
I found a problem in the window of Rectangular Domain Discretization. I manually entered the horizontal and vertical discretizations, and of course the mesh for a rectangular domain was generated as defined. When I closed and reopened the project, the values in the rectangular domain discretization window changed back to the default liner values, although the mesh in the graphical window (the center of the main window) was still remained as manually defined. This also occurred when I copied the project.
I think this disagreement will be a problem in editing the discretizations. Is this a bug in program?

Best regard,
msakai

 Mirek11/23/2006
Hello,
I tested the dialog in version BETA.07 and it seemed to be working correctly. Horizontal and vertical discretization values were saved and reloaded correctly. If you use also version BETA.07 could you send me the project? If you use an older version could you test it in version BETA.07?

Regards Mirek


msakai11/24/2006
Hello, Mirek.
I was using Beta.06, and now I tested in Beta.07.
Beta.07 works correctly as you say.
Thank you for your help.
msakai


## ID = 714, Volumetric flux in = volumetric flux out?

Brian11/30/2006
Good Day,
I have discovered a discrepancy of sorts in the water balance of a 3D water flow model. The model is an elongated cube, with a constant flux boundary at the top surface where 0.537m^3/d of water is entering the model through 99 nodes. This input is mimicking a treated wastewater drainfield. The long sides and bottom of the model are no flux boundaries, and the short sides at either end are constant head boundaries that are set to the initial conditions at each end of the model. The model domain is 1.5 m of sand overlying 3.5 m of sandy loam, with the upper (approximately) 3.5 m of the model being unsaturated. The water table is initially angled at -0.2034 degrees from left to right.

The water input rate into the model through the constant flux boundary at the top surface is sufficient to cause a water table mound to form in the middle of the model, with flow subsequently exiting the model at both the left- and right-side constant head boundaries. Since the model quickly reaches steady-state, I expected that if I multiplied the saturated cross-section (1.5 m^2 at the right side, 1.8 m^2 at the left side) at each of these two boundaries by the saturated hydraulic conductivity of the sandy loam (0.59 m/d), multiplied each of these values by the hydraulic gradient, and divided each of these values by the porosity (0.41), and then added the two results together, that I would obtain a volumetric flux value approximately equal to the flux entering the model at the top (0.537 m^3/d). However, I am surprised that this is not the case: I obtained a value of 0.193 m^3/d. I noted that the hydraulic gradient reached a maximum at the either edge of the model, and I determined the maximum gradient value for both the left and right sides of the model using the results cross-section tool, but even using these gradient values (0.05 at the right side, 0.035 at the left side), the volumetric flux is only 0.193 m^3/d.

There seem to be two potential reasons for this discrepancy:

1. I am not accounting for unsaturated zone flow out of the model, just above the water table (true, but I don't think that this accounts for the remaining 0.344 m^3/d of volumetric flux exiting the model.)

2. Perhaps the hydraulic gradient at each edge of the model is greater than it is shown. For example, the maximum hydraulic gradient shown via the results cross-section tool

is 0.05. Perhaps I do not understand how Hydrus calculates hydraulic gradient from one side of a constant head boundary (in the model) to just outside the model. In order for model water influx to = model water outflux, the hydraulic gradient at either end of the model would need to equal approximately 0.15, which is not unreasonable, but is not shown in the model output. How can I determine if this is the problem? I do note that the water velocity is greatest at either edge of the model, suggesting that the hydraulic gradient is increasing at toward the constant head boundaries. How is hydraulic gradient calculated across the constant head boundaries at these model surfaces?

Thanks,
Brian


Jirka12/03/2006
Brian,
You can see how the gradient is calculated in the code from equation 5.28 and 5.29 in the Technical Manual.

You could see best whether the outflow flux is equal to the inflow flux from GUI graphs of the actual boundary fluxes (in the v_mean.out file) or by looking at the mass balance (in balance.out). If you want to separate fluxes at the left and right side, you need to use different BCs on the left and right. You can use constant head BC on the left and variable head BC (even with constant head in time) on the right. Then the code will calculate fluxes at both sides separately. You can also calculate the fluxes at each side by integrating fluxes at individual nodes (e.g., in Excel) on the left and right boundaries, which are given in the Boundary.out file. I would be surprised if the inflow was not equal to outflow at steady-state. If it indeed is, please, send me the project where you can see this discrepancy.

Best regards,
Jirka


Tim, 12/04/2006
Brian,
Hope it's okay if I jump in here. I think you have the volumetric calculation wrong; you should not be dividing by porosity. $Q = KiA$ where $K$ is hydraulic conductivity, $i$ is gradient (steady), and $A$ is cross-sectional area (perpendicular to gradient). You only divide by porosity in the Darcy equation to estimate average linear velocity.

Unless I misunderstand, you have described an essentially 2D flow problem; why use a 3D model? Is the extra computational effort rewarded with some benefit? If the drainfield is approximately circular or square, you can use an axisymmetric model; if it is linear you can usually use a slice perpendicular to the alignment.

Check to make sure your boundaries are not too close to the recharge nodes. Gradient will always increase as you move away from the mound (because the cross-sectional area of flow is decreasing), but if the BC is too close there will be an abrupt increase

in gradient near the boundary. If the gradient decreases smoothly, proportionate with the saturated thickness, then you're probably OK.

Hope this helps,
Tim


Brian, 12/04/2006
Hi Jirka and Tim,
Thanks for your comments.
Tim, I agree with you completely regarding using a 2D model for this scenario. This particular example is from a previous model run where I was still using a 3D model with, as you state, there being no advantage over a 2D model.

You are also correct in that the recharge nodes are probably too close to the constant head boundaries located at either end of the model. I am also wondering about something else: the use of constant head boundaries at either end of the model (the yz planes). By using constant head boundaries, the additional flow resulting from recharge from the modeled drainfield is forced through the same cross-sectional area as existed before the drainfield began recharging water to the subsurface. Thus, although a water table mound will form beneath the center of the drainfield, once this additional flow approaches the constant head boundaries it is forced into a constant cross-sectional area, which was defined by the saturated thickness at the ends of the model set by the water table at time = 0. Keeping the cross-sectional area constant results in the hydraulic gradient reaching a maximum at these two ends of the model: referring to Q=KiA, since A is smallest there, i is greatest. Thus the flow from the drainfield cannot increase the saturated thickness at the ends of the model if I use constant head BCs.

It seems that this scenario is not realistic. In reality, the water table should increase in height not only at the center of the model but also at the ends of the model (as long as the ends are located within some proximity to the recharge zone, which they are in my model). By using the constant head boundary BC at these ends, I am not allowing this to occur. Is there a different BC that I should use at those two locations to allow for a water table height increase? How "far away" from the drainfield should I located the constant head (or other type) boundaries?

On the other hand, perhaps using constant head BCs at the ends is a conservative approach because perhaps this forces groundwater mounding at the center of the model to occur to the maximum degree possible.

I would really appreciate your opinions on this.
Thanks much,
Brian
**ID = 715, Sorbed concentration – initial conditions**

Eatkinso, 11/30/2006
Hello,
I am fairly new to Hydrus 1D and I am trying to model some desorption experiments. As I understand it, I should be able to specify the initial sorbed concentration within

the Soil Profile Editor. However, when I attempt to select the 'sorbed concentration' from the initial conditions drop-down menu, I am not able to select it (the words are gray, rather than black).

Why am I not being allowed to specify a sorbed concentration? Could it have to do with my boundary conditions??

Any help would be extremely helpful.

Thanks,

Erin

ntoride12/02/2006

Erin: The sorbed initial concentration must be specified only when the nonequilibrium adsorption is considered (see Eq.(3.23) in HYDRUS-1D manual). For the equilibrium adsorption, the sorbed initial concentration is determined based on the initial solute oncentration and the adsorption isotherm. Nobuo

Jirka12/03/2006

One more comment: for the nonequilibrium adsorption to be considered, the Frac coefficient (fraction of sorption sites in equilibrium with the liquid phase, the remainder is kinetic) must be different from one.

Jirka

## ID = 716, CXTFIT one sitemodel

Wanlutfi,  12/01/2006

Hi. I'm using Stanmod 2.2 CXTFIT. The problem that I encountered was when trying to use first type inlet, Cr and set it as constant initial concentration, I received these messages:

WARNING ! Z0 > 0 FOR A FIRST TYPE INLET

WARNING ! Please check transport parameters.

Any help would be appreciated.

Thanks

Wan

ntoride12/02/2006

Wan:

Since the first-type BC specifies the surface concentration, the message was originally designed for the delta initial condition. Although I corrected this problem, the old cxtfit2.exe was probably accidentally used for Stanmod 2.2. I will put the correct one at my university web site on Monday. We will also correct the Stanmod file.

Please also note that the difference between the flux concentration and the resident concentration subject to the first-type inlet condition (Toride et al., SSSA, 1405-1409, 1993).

Nobuo

## ID = 717, CXTFIT2.exe (corrected address)

ntoride12/04/2006
Dear Stanmod users:
Please update cxtfit2.exe (2000/10/30 -> 2005/2/16) in the Stanmod
folder for some corrections including the first-type inlet condition
problem.
http://www.bio.mie-u.ac.jp/junkan/bus****su/lab5/NT/cxtfit/cxtfit2.zip
Please use the alternative domain address:
http://www.bio.mie-u.ac.jp/junkan/sec1/lab5/NT/cxtfit/cxtfit2.zip
I will delete the file at my personal site soon
(http://homepage.mac.com/wanny/NT/cxtfit2.zip)

ntoride12/04/2006
The web-site address did not appear correctly. The correct one is
http://www.bio.mie-u.ac.jp/junkan/bus****su/lab5/NT/cxtfit/cxtfit2.zip
If I fail again, please replace under /junkan/ as follows:
bus****su/lab5/NT/cxtfit/cxtfit2.zip
Nobuo

Ntoride,12/04/2006
Sorry for the trouble. This forum seems not to allow some combination of the
Japanese word. Please try this site:
http://homepage.mac.com/wanny/NT/cxtfit2.zip
Nobuo

wanlutfi12/05/2006
Thanks Nobuo.

## ID = 719, Reactive solute transport

nasserga12/16/2006
My Dear Kirka
When I simulate a reactive solute such as Cd, if I need to know the sorbed fraction of
Cd, where can I found these values? in solute 1.out or conc.out files or it is in other
file.
Thank you
Dr. Gamal Khalil

Jirka, 12/27/2006
Gamal,

The sorbed fraction of solute is printed to the Nod_inf.out file, but only when the kinetic sorption is considered. If equilibrium sorption is considered, then this information is not printed, since it can be easily calculated from the sorption isotherm, i.e. for the linear sorption isotherm as Sorbed concentration = dissolved concentration (Conc) * Kd.

The amount of solute in equilibrium phases, i.e., liquid phase and sorbed instantaneously is also printed into the balance.out file.

Best regards,
Jirka


## ID = 720, Time variable heat transport boundary

Gijsbert12/19/2006
Hi
I'm trying to model heat transport from an infiltration channel to an other lower situated channel in Hydrus 2D. Both channels have constant pressure boundary conditions. The surface of the dune area between the channels is given an atmosferic boundary condition. Water flow is working correctly but when I try to model heat transport I got some problems. For the heat transport i want to give time variable boundary conditions to both the soil surface and the infiltration channel. For the soil surface with the atmosferic boundary condition it is working but when I try to give the channel (with constant pressure boundary) a time variable boundary condition it is overruled by the constant value given at Tbound2 in the heat transport parameters dialog box (in my case the value is set to zero).

It looks like it is not possible to give a time variable heat boundary condition to a constant pressure boundary, can anyone give me some clues on this subject? Should be possible i think because it is usefull for many applications

Regards,
Gijsbert

Jirka12/28/2006
Guisbert,
You can not indeed apply the variable temperature on the constant pressure head BC. Constant head and flux BCs can be associated only with values from the tBound vector. If you want to change temperature on this boundary, you can use time-variable head BC (even with constant head) on this boundary and associate it with TValue2 (while atmospheric BC can use TValue1).
Jirka

Gijsbert  01/08/2007
Hi Jirka
Thanks for your reply
I figured it out myself already and it works ok
Regards Gijsbert

**ID = 721, Hydraulic conductivity**

Srilert12/28/2006
Dear all,
I have a question that I want to use soil hydraulic properties(e.g. water content, hydraulic conductivity etc.,) when I have already run the HYDRUS-2D. I know I get this data from "check.out" but when I plot this data between water content and pressure head I found that the curve was not showed the residual water content, for example, I input saturated water content = 0.456 and residual water content = 0.10 . when I already run HYDRUS-2D and I got the data from "check.out" and I could plot some part of water retention curve only (from water content =0.45 to 0.19) I cannot plot the tail of this curve.
Could anyone suggest me for this problem ?
Thank you for your assistance and merry Christmas and Happy new year.
Srilert


Jirka12/28/2006
Lert
HYDRUS draws the data only between values specified for the Internal Interpolation Table. If this table is disabled, then the code draws graphs for pressure heads between -0.0001 and 100 m. If you want to have figures of soil hydraulic functions for wider intervals, you need to specify them. You can also use other software, such as RETC to do these graphs.
Jirka


**ID = 722, Acutal vs potential atmospheric flux**

rbeggs12/28/2006
I am running a 2D model with beta 0.7 and the atmospheric flux results don't seem to be correct. Precipitation is showing up in "Potential Atmospheric Flux" under the Boundary Fluxes option in results, but it is not showing up in Actual Atmospheric Flux nor in the Cumulative Flux displays. It looks like it is entering the domain correctly based on the water content results. However, in the v_mean.out file, all the rAtm is negative and offset by positive values for Runoff, leaving no vAtm.

A separate problem I am also having is that when the water table rises to the surface (atmospheric BC), ORTHOMIN errors cause the program to abort rather than letting the excess water leave the domain.


Jirka12/31/2006
In HYDRUS, the fluxes in v_mean.out and CumQ.out files that are directed out of the domain are positive, and those that are directed into the domain are negative. That's why rAtm are displayed as negative. Why there is no infiltration from precipitation? Have you assigned atmospheric BC? Are the pressures at the atmospheric BC

negative (i.e., it is unsaturated)? Do you saturate the entire transport domain? Other than that I do not know what to suggest. Should I look at your project?
Jirka

rbeggs01/02/2007
According to the v_mean.out file, there is no infiltration from precipitation. But it appears to be showing up in the moisture content. Atmospheric BC was assigned to the top. A small part of the top surface does saturate at a few time steps. I will send the project files.

Jirka01/04/2007
Rob,
There was indeed a problem in the code for the combination of having atmospheric BC, head interpolation in time, and switch from time-variable to zero flux BC. I have fixed that and updated the code (I have emailed you the update).
Thanks for discovering this problem and informing us about it.
J.

## ID = 724, Use of CDAD for simple GW transport

Mvcallaghan, 01/09/2007
Hi All,
I have been testing 3DADE for simple 1D groundwater transport of chloride. The way that the source dimensions are set out in the program are for vertical transport in a one dimensional flow field. Is there any conceptual problem with rotating the frame of reference so that the flow is modeled in a horizontal direction? The rotation is simply conceptual and easily done; however, I'm wondering if there might be some underlying concepts in 3DADE which preclude it's use for horizontal transport.

Does anyone have an opinion on this?
And what other potential pitfalls are there to using 3DADE for simple GW modeling? For example, since the model is analytical I am assuming there are no mass balance issues to worry about. Is this a correct assumption?

Thanks,
Mike

Mvcallaghan 01/09/2007
I have since found many examples of the use of 3DADE to horizontal GW transport. Including its implementation in the SOLUTRANS program. So I guess that I've answered my own question...yes, 3DADE can be used to simulate 1D horizontal groundwater transport with 3D dispersion.
Mike

**ID = 725, Calculating of groundwater level**

Brano,  01/10/2007
I have question about hydrus1d/2d.
I would like to simulate (calculate) groundwater level changes/movement in wetland area. I was looking in hydrus demos and manuals but without any result. I dont have any inputs data from both sides left and right (some groundwater inflow or outflow or some ditch or river on the side), only atmospheric datas, plant datas, soil datas and groundwater levels datas. So I would like to consider with vertical direction of flow. I imagine that I put all my datas (atmospheric,soil and initial value of grounwater level) to hydrus 1d/2d and the result will be the movement (changes) of groundwater level (table) in time. So I would like to compare measured and simulated groundwater level. I find in Example2 - Grass Field Problem that you put initial value to model but in demo I couldn't find it, where is that value. I dont know if this is possible with hydrus1d/2d or some other mathematical model, but I would appreciate it if you could help me with this problem. Thank you.

Jirka01/10/2007
HYDRUS codes simulate flow in both saturated and unsaturated zones simultaneously. The pressure heads in the saturated zone are positive and in the unsaturated zone negative. Groundwater table is located where the pressure head is equal to zero. Initial pressure heads are given as initial conditions. Thus you need to look into the initical condition for the location of the groundwater (it is 55 cm below the soil surface.
J.

**ID = 726, PDF document with all Hydrus-2D discussion topics**

 Mirek, 01/10/2007
Dr. Liping Pang and her students have created a PDF document with all Hydrus-2D discussion topics. The document can be downloaded here: Discussion_H2D.pdf (1.13MB) or at the main page of our discussion forums: http://www.pc-progress.cz/_forum. Thank you Liping!

**ID = 728, Boundary condition for irrigation optimization**

marisun01/12/2007
Hello,
i have a problem with the decision for an BC.The time for an optimal irrigation is searched. I used variable flux with boundary condition and explained a time-function (10mm/h)for the rain. Than i used the constant flux BC with the same Flux rate per time step like in the time function. The results are very different!!! (1. 108hours, 2. 35hours) what is the problem?? which BC is the right one?? I can not understand what is explained in the manual...i hope, anybody could help!! i would very much

appreciate.

Thanks, Mari

## ID = 729, Printing times

Ekkehart, 01/12/2007
Hello,
I would like to force Hydrus to print results for the boundary conditions (e.g. cumulative fluxes) just for predefined print times. When I'm switching off the toggle for T-Level information and interval output Hydrus is using my defined print times but not all of them (only later than 780 seconds). Does anybody knows the reason for this? Thank you very much in advance.
Yours Ekkehart

Jirka, 01/12/2007
It is possible that you do not allow the code enough flexibility in determining time steps and it just can not hit your early print times. Make sure that your initial time step and minimum allowed times steps are small enough (order of magnitude smaller than the first printime, or time difference between other print times) so that your required print times can be hit during time stepping.

Jirka

Ekkehart, 01/14/2007
Dear Jirka,
thank you very much for your reply. Although the minimum and initial time step length is set to very small value (e.g. 0.01 sec) Hydrus2D/3D is skipping the first some hundered seconds before printing out values for the print times. In Hydrus1D this is working without any problems.
Yours Ekkehart

Jirka, 01/14/2007
You need to send me the Project_name.h3d file, otherwise I can not comment. J.

## ID = 730, PhD thesis defence – HYDRUS simulations

Julie01/16/2007
Dear all,
You are cordially invited to the defence of my thesis named :

Water flow and transport of potassium and nitrate ions in a variable charge soil under rainfall redistributed by crop
Experimental and modeling study in a banana plantation fertilized on an andosol

The defence will take place in the Risler room of the Agronomic National Institute of Paris Grignon (France) on January, 30, 2007 at 2.30 PM in front of the jury composed by :

M.Alain Perrier,Professor, INA-PG, President

M.Bruno Delvaux, Professor,Catholic University of Louvain, Examinor

M.Roger Moussa, Research Director, INRA, Montpelier,Examinor

M.Philippe Cattan, Researcher, CIRAD, Guadeloupe, Reviewer

M.Cyril Kao, Researcher, CEMAGREF, Anthony, Reviewer

M.Fabien Thomas, Research Director, CNRS, Nancy, Reviewer

M.Yves-Marie Cabidoche, Research Director, INRA, Guadeloupe,Director

The summary of the thesis is :

Water flow and ion transport in the vadose zone of an andosol under banana plants are complex processes controlled by hydrodynamic and chemical mechanisms active at different spatial scales. On the one hand, the banana plant intercepts the incident rainfall and concentrates it more than 20 times around its pseudo-trunk (stemflow). On the other hand, soluble manure manually and intensively spread near the plant foot is exposed to this stemflow and carried away to depths of the variable charge soil that simultaneously sorbs both cations and anions.

These characteristics make banana plantations on andosol very complex. Decisions about configurations on which to carry diagnosis or about optimized fertilization practices can be made only if one understands and can envisage outgoing flows of aqueous solutions. To better understand and manage such systems, laboratory experiments (measurements of the soil hydraulic properties using the Wind evaporation method; batch experiments, and soil columns experiments), in situ experiments (measurements of the hydrodynamic parameters using the Beerkan, Decagon, Trims, double rings infiltrometer methods, lysimetry with controlled suction, and tensiometry) and analyses with the numerical HYDRUS-1D/2D/3D models were carried out.

The objectives of the thesis are (i) to measure and model water flow at the scale of the plant and its row, (ii) to measure and analyze the behavior of the $K^+$ and $NO_3^-$ ions at batch, column and field scales, and (iii) to model the water flow and ions transfers at the scale of the soil column.

The adsorption isotherms established in batch systems showed the capacity of the B horizon, under acid conditions, to preferentially adsorb nitrates compared to potassium. Opposite behavior was found for the A horizon. Measurements on soil columns confirmed this behavior. Fitted parameters of the linear and Langmuir-Freundlich isotherms provided satisfactory model descriptions of solute transport in one-dimensional column experiments. In situ, results showed that the main stemflow impact is directly under the banana plant and immediately downstream of it. The soil hydraulic parameters (parameters of the retention curve and the hydraulic conductivity function), obtained using an inverse procedure that considered as objective function the lysimeter fluxes, were in good agreement with saturated hydraulic conductivities measured using double ring infiltrometers. Two- and three-dimensional water flow modeling was validated by comparison with the tensiometric data. The similarity of the simulated fluxes obtained with and without lysimeters

made it possible to show that the lysimeter device did not affect results during studied periods. The water balance for the banana plant and the row and duration of 168 days, is reasonable, with an error of 4%, despite of a considerable rainfall of 4120 mm during the studied period. Solute transport modeling at the plant scale was not done since the large water fluxes did not permit in situ equilibrium to be reached. However, the experiments showed that the stemflow considerably affected the potassium and nitric nitrogen losses when manure was spread at the foot of the banana plant. Ions then did not have enough time to interact with the anion and cation exchange capacities of the soil and were quickly leached. The stemflow affected the retention of nitrates and potassium less if fertilizers were spread uniformly on the soil surface. In spite of very high rainfalls typical for regions with andosols, these soils are able to prevent the leaching of abundant nitrates under excessive fertilization, except when manure is exposed to localized drainage fluxes, such as stemflow resulting from the rainfall focused by the cultivated crops.

Key words: stemflow, spatial distribution of drainage, modeling, water flow, variable charge, anion and cation exchange capacities, banana plant, andosol, potassium, nitrate.

It will be a pleasure to invite you for a drink after the defence.
Regards,
Julie Sansoulet


Jirka, 01/16/2007
Julie,
I would like to congratulate you on finishing your thesis. I believe that you are the first PhD student who was using the entire suite of our programs, including our new HYDRUS (2/3D) software. I hope that your experience with our software was only a pleasant one. I'm sure that at least the trip to Barbados to take our HYDRUS short course must have been. I hope that you will keep on using our software in the future as well.

Good luck during the defense.
Jirka

PS1: You need to defend against one president, one director, two examiners (make sure that do not drag you to those infamous CIA secret examination prisons in Eastern Europe) and three reviewers. One woman against seven men! That should be a piece of cake, right?

PS2: If you want to further deepen your knowledge of our software I will recommend you upcoming short courses in Hanover (Germany), March 23-25 or Shrewsbury (England), March 27-29.

PS3: I have wonderful memories on my summer stay in Grignon in 2003. Perhaps you will have the same on winter 2007.


Julie01/16/2007

Jirka,

Thank you for your congratulations.

Of course, my experience with HYDRUS, whatever the dimension, was full of personal enrichment. I will certainly keep on using your software in the future : I am a new worker of the International Center of Agronomic Research for the Development (CIRAD) and I am going to study water and solute transport (again). This time I am not in the french West Indies (caribean area) but in "La Réunion" (next to Madagascar), which is also a kind of ex-colony. I am thinking about using Hydrus 1D and HP1 this time.

Best regards,
Julie
PS1 : no worry, we are puppets in a theatre.
PS2 : what about a hp1 course ?
PS3 : This part of "National Institute of Agronomy - Paris Grignon" is in the center of Paris, 16 rue Claude Bernard, Paris 5, métro : Censier Daubenton (line 7).

Hector, 01/30/2007
Chere Julie,
for sure there are a bunch of good reasons to go to Paris...
Fingers crossed & good luck for your defense!

Hector

## ID = 731, Simulation of solute injection

Nasserga, 01/17/2007
My Dear Prof. jirka
I have a research study for simulating the irrigation practice , if we need to simulate the solute injection for 1 hr at the begening of 5 hrs irrigation period(a hr solute+water then 4 hrs water only). I try to simulate this case with time -variable boundary conditions, but the results are not logical, then what is the problem?.

please inform me the correct method to express my case study.
Thank you so much
Gamal Khalil

Jirka01/17/2007
Gamal,
I would guess that you must have something wrong in your input. I have been simulating much more complex conditions, e.g., first only water for 20 minutes, then fertigation for an hour, and then water again, and it was working fine. See the reference given below.

Gärdenäs, A., Hopmans, J. W., B. R. Hanson, and J. Šimùnek, Two-dimensional modeling of nitrate leaching for various fertigation scenarios under micro-irrigation, Agric. Water Management, 74, 219-242, 2005.

Check again your input, mainly the time-variable BCs.
Jirka


## ID = 732, Drainage

Andrea, 01/23/2007
Dear Jirka,
I am new to use hydrus and i am trying to simulate the discharge of a chemical in groundwater.
I selected a value of 300 cm as depth of the soil profile.
I considered that also the pressure head was at 300 cm of depth, at the end of my soil profile.
Now i want to know if is possible to select as lower boundary condition a free drainage(zero gradient); can i do it? has it a sense? I tried to do it,but not all the results converged.
Instead, if I select a costant pressure head the results converged. Why this difference?

Thank you very much!


Jirka, 01/23/2007
If there is a groundwater at the bottom of the profile, you should use constant BC with zero pressure head.
 J.


Andrea, 01/24/2007
Daer Jirka,
I selected a costant flux of -5 cm/hour as upper boundary condition, but i selected a value of pressure head distrution(in soil profile )of -300 at top and bottom.
Is it wrong in my case? Where I must select zero pressure head?
And what i must select as lower buondary condition?
Thank you you very much!


Jirka, 01/24/2007
Andrea,
I can not help you with boundary conditions. You need to think about what your physical system is about and accordingly specify boundary conditions. Boundary conditions have to reflect your system.
J.

## ID = 733, Breakthrough curve bromide (Hydrus 1D)

Andre, 01/28/2007
Hello,
I study Geography and Chemistry and I am currently working on soil column experiments (Tracer studies/100% sand soil).

I created a breaktthrough curve of my bromide-tracer (concentration = 9,89mg/l) and now I'd like to compare this curve to a computer-simulated curve (created with Hydrus 1D). I just do not know who to use the program. The main problem is that I don't know which parameters I have to enter in this programm. Which options do I have to activate? Here are the informations I have:

thetas 0,5442 Vol.-%
thetar 0,0508 Vol.-%
alpha 0,0564 -
n 2,4905 -
m 0,60 -
Ks 616,46 cm/d
ku 17,28 cm/d

length of the soil column: 30cm
pressure at the bottom of the soil: -25mbar/hPa (compared to air pressure)
soil surface: 70,85cm2
flowrate (tracer): 13,2 ml/h (impulse application for 24hours)
unsaturated conditions in 100% sand soil

I would be very happy if someone could help me.
Best regards
André


Jirka, 01/29/2007
Check out the example that come with HYDRUS-1D. Mainly Test4 and Test5 for direct problem and FitTest4 and FitTest5 for inverse problem. That should get you started.
Jirka


## ID = 734, Interflow simulation

linlin01/31/2007
hello1
I want to simulate the interflow under heavy strom about 14cm/d,the mmodelling region is a slop field,that is about 160m in length and 3m in depth.there are three soil layers,and the deeper soil depth is,the less Ks is.
however,the processes ceased when caculated with the such heavy rainfall.
do you know how to solve the problem!


Jirka02/01/2007
It is likely that you need finer time and space discretization. Start with smaller transport domain and see what works, and only then simulate the entire transport domain.

Jirka

**ID = 735, Hydrus for leaching of N and P**

Luca, 01/31/2007
Hello
please I need help.
I'm searching for a model which could allow me to simulate the transport processes in a variably saturated soil of NO3, NH4 and dissolved P forms after the application of liquid hog manure. The model should also be able to model chemical processes related to nitrogen (mainly redox potential leading to nitrification/denitrification) and phosphorus (mainly sorption/desorption processes).

Is this Hydrus-1D-Phreeqc good for this?
thank you

Jirka02/01/2007
Luca,
I would initially use just HYDRUS-1D itself. It has option for sorption, nitrification and denitrification. Only once that is not sufficient in describing all the processes then I would go to HP1. HP1 is not a simple program to use and it does require a prior knowledge of PHREEQC.

Jirka

**ID = 736, BBC – deep drainage**

Brano, 02/01/2007
Dear Jirka,
how did you set Aqh and Bqh parameters in q(GWL)- relationship in deep drainage bottom condition?

Jirka, 02/01/2007
See the answer in the topic 566, which deals with this issue.
J.
http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=566

**ID = 737, PHREEQC – tutorial?**

Uwe02/01/2007
hi,
anybody know an online tutorial for PHREEQC? I have lots of experience in hydrological/hydrogeological modelling, but the fist steps with PHREEQC are rather difficult...

uwe

diederik02/01/2007
Uwe,
there is some information on the web:
a phreeqc course on : http://wwwbrr.cr.usgs.gov/projects/GWC_coupled/phreeqc/
Tony Appelo's homepage and training site:
http://www.xs4all.nl/~appt/index.html
you find information and some example input files
plus a 15 page condensed manual (phreeqc downloads)
on http://www.sckcen.be/hp1/ you can download some detailed instruction to prepare input files (see HP1 course)

greetings,
diederik

### ID = 738, Two dimension dispersion equation

Srilert02/03/2007
Dear all,
Could anyone help me to find two dimensional equation for solving longitudinal and transversal disperion coeficient of continuoes point injection ? And If I would like to find DL AND Dt from experimental data by inverse module in Hydrus 2D, How should I do ?

Best regards,
Srilert

### ID = 739, Editing data

jsprice02/08/2007
I'm a new user - sorry if this is dumb. I want to use my own soil profile information. The edit-data command (right click on graph) brings up a spreadsheet but I cannot cut and paste from Excel, for example (it just returns a "xero" in the one cell I paste to. There must be a way to do this. There doesn't seem to be a ready guide to finding the data in the approriate text file either. Hints?

Jirka02/08/2007
Input to the model is on the left side of the main window. You are trying to edit the results (which are on the right side), which is obviously not allowed.
Jirka

**ID = 740, Flux**

andrea02/08/2007
Dear Jirka,
what dimensions has the upper boundary flux?
I don't find this information using the Help option.
Thank you very much!

Jirka, 02/08/2007
Lengh/Time, e.g., cm/d depending on units you select.
J.

**ID = 741, Air entry – 2cm (size counts)**

jsprice02/12/2007
Hi. I am running Hydrus1d in a soil with large pores (moss actually) for which I've done the retention, Ksat and Kunsat so have VG parameters. Using an upward flux of .05 cm/h, head constant at zero w at z=-30 cm. When using air-entry of -2cm the profiles seem reasonable - surface heads >=-200 cm, initial flux at surface .05 cm/h, eventuly reaching this throughout the profile after ~8 hours. Switch +2 cm option off and heads will go to -10 000 or whatever I'd let them, flux drops to zero a few cm below surface. Since the profile starts at equilibrium pressure (0 cm at base, -30 cm surface)and therefore unsaturated, why does the +2cm make such a difference? Thanks for any insight you can provide.

Jirka02/12/2007
The air-entry value affects quite dramatically unsaturated hydraulic conductivities. Thus if you remove that option the conductivity of the soil (for -30 cm) may drop orders of magnitude (depending on other parameters; see the graphs of your project). Thus if you specify a surface BC as a constant flux (which I guess you do), then the soil may not be able to deliver water at that rate and the solutions does not converge. Instead of using a constant flux, use atmopsheric BC with evaporation equal to your flux, and the code will likely find the solution, since it is more flexible for the atmospheric BC than for constant flux BC (where it has no flexibility, it enforces the specified flux no matter what; until it crashes).
J.

jsprice02/12/2007
Jirka - you are correct - I used the specified flux for evaporation of 0.05 cm/h. But it did not crash - I got profiles but the flux very quickly dropped and equilibrates around 0.002cm/h throughout the profile. I was surprised since I was expecting it to be forced to 0.05 cm/h. I changed the upper boundary flux to Evap and the result is essentially the same.

I know the evap can actually be 0.05 cm/h. THe only way I can get this as a surface flux is if I play with the retention curves to keep more water in the soil. However, this doesn't agree with my measured curves.

I have a set of retention and Kunsat curves in Hydrus that look close to the measured ones. The boundary conditions are reasonable. The output is not. Maybe it indicates that groundwater flow processes in moss matix (very high saturation vmc=0.9; very high Ksat; poor retention and low Kunsat) doesn't occur as we excected it to (they are non-vascular i.e. transport only through capillary flow nothin internal). Active porosity is substantially lower (~50%) so maybe the dual poroaity function is needed. Frustrating but interesting.


Jirka02/12/2007
Small air-entry value (i.e., -2 cm) has almost no practical effect on retention curve, but substantial effect on conductivity. I still think that by using this option, you must have increased substantially unsaturated conductivity. You can achieve similar effect by decreasing l (even negative). Then the soil will be able to delifer water toward the surface.

The dual-porosity and dual-permeability models are meant mainly for infiltration, when preferential flow paths become active. For upwared cappilary rise there are less applicable (although I have disproved this statement in this paper: Simunek, J., O. Wendroth, N. Wypler, and M. Th. van Genuchten, Nonequilibrium water flow characterized from an upward infiltration experiment, European J. of Soil Sci., 52(1), 13-24, 2001.).
Jirka


jsprice02/13/2007
Yes - decreasing L increases K, but the modelled K values then don't agree with the measured values.
Since the model is not producing the specified flux at the surface (no matter if I release the minimum pressure head), does that mean the model is not operating correctly? I understand why it does not when I use evaporation (which depends on water being available), but when the flux was specified, and the flux was not achieved (no indication of a crash), does that mean the solution is not found (I see no message saying such)? Your comments are very helpful.


Jirka02/13/2007
I would need to see the input files (you can zip it and email to me) to comment on why it does not reach a specified flux.
J.


Jirka02/14/2007
Jonathan,
I have looked at the files you sent me. In that project you specified the upper BC as atmospheric boundary with evaporation flux. HYDRUS worked exactly as it should.

It delivered evaporative flux until the limiting surface head was reached, which occurred at about 0.4 h. Then it switched BC from flux BC to head BC (limiting head) and calculated the decreasing evaporation flux. The only problem I saw was that you specified the limiting pressure head at the soil surface equal to -100 cm, while it should be on the order of -15,000 cm or even lower.

You did not send me project with specified flux. Thus changed the BC in your project to specified flux. HYDRUS worked again exactly as expected. It delivered the flux as long as it could (which was again about 0.45 h) and then the surface pressure went below reasonable values and the code crashed. This indicates that the project is not physically well defined, since the specified BC can not be fulfilled. One clearly needs to use atmospheric BC for this type of problem.

Jirka


## ID = 742, Other hydraulic models

Franta, 02/13/2007
I am using Hydrus2D to inversely estimate soil hydraulic properties for layered soil from measured tile drain flux, heads, a TDR water contents. I have had some success, but would like to try some other hydraulic model, i.e. dual permeability. However, this model is "inactive" in our version 2.102. I also downloaded the Beta version of Hydrus3D and briefly looked thru the available models and again the dual permeability is not available.
Is there a way to implement the dual permeability model in our version or do we need to purchase upgrade to the 3D version?

Thank you
Franta


Rvang, 02/13/2007
Franta:
You can use the "Modified VG" model (see soil hydraulic model), and do an inverse with that model. The modified VG adds a linear extension to the K(h) model to account for macroporosity. This is discussed on page 17 and 18 of the older HYDRUS2D manual (Fig. 2.3).
Rien van G.


Jirka02/13/2007
Franta,
Dual permeability model is a rather complicated model (both conceptually and numerically), and although we have already implemented it in the computational 2D module of the HYDRUS (2D/3D) software package, we have not made this option so far available to general users. I would recommend you to first try, in addition to the model suggested by Rien, the dual-porosity models that are available in HYDRUS (2D/3D) and that are much simpler to interpret.

Jirka

**ID = 743, Changing print defaults**

Rbwhite, 02/15/2007
I would like to change the defaults for plotting the graphs that occur as output in the post-processor. I have found a way to make that change manually on each individual graph, but would like to change the default to make global modifications on such things as the line thickness, size of the graph on the page, etc. How can I make changes to the default options?
Rich

Jirka02/15/2007
HYDRUS-1D does not have an option to save these modifications globaly. They are kept in memory only during one graph display sequence. We realize that this should be a nice option to be able to save it globally also for other projects and have added this option recently to HYDRUS (2D/3D). We will likely add it in the future also to HYDRUS-1D.
Jirka

rbwhite02/15/2007
Thanks for the feedback. You may want to think anout this in a future version. In all other respects, this is a great model.
Rich

**ID = 744, Root uptake**

jsprice02/16/2007
I am using the root water uptake function. I have set it to 0.03 cm/h. The evaporation is set as the same. However, the root water uptake seems to make no difference. I have the parameters set so transpiration is supposed to operate anytime pressur is <0 and at maximum rate at P<0. My pressure never reaches P2L but still there is not root flux. Sugestions?

Jirka02/16/2007
Have you specified root distribution with depth?
J.

**ID = 745, Atmospheric boundary**

onurakay02/16/2007

Hi. I have just started using Hydrus3d package. You may find some new topics from me for the next couple of weeks. I'll pay attention not to post already solved issues. Any comments will be appreciated.

Now, my first issue: I am trying to model a lab study that involves a soil column with a 1 ft diameter and 2 ft long. My problem is that although I am putting high precipitation rates in Atmosph.in (considering the Ksat value of the material I am using) the pressure head could rise only to zero at that top boundary. I changed the 'hCritS' to 50 cm in the input file. This time only the initial pressure head changed to 50 cm at the top which wasn't my intention. The model after the initial time started to give 0 head again. As you can guess, I wasn't getting a perfect mass balance. The water amount from my rainfall hydrograph was't going into the system when I compare this amount to the cumulative input at the atmospheric boundary.The actual cumulative value wass lower. There was no evaporation.

Secondly, I lowered my precipitation rates to see if the model is putting the total amount of water into the system. But again the total amount went into the system was lower. What can be the reason?

Thanks.

Jirka, 02/16/2007
Both 2D and 3D codes assume that any excess water from pricipitation that does not infiltrate is removed instantaneously by surface runoff. Thus you should not get pressure head values at the atmospheric BC larger than zero.

Jirka
PS: Remind me, where do you enter in GUI hCritS.

onurakay02/16/2007
Dr.Simunek,
Thanks for a rapid response and information. According to the technical manual, it says the hCritS is usually set to zero. Since only the minimum pressure head value can be changed via GUI, I had to go in the input file to change it from there. Do you have any comment on the total volume not being put into the model? I tried to change the time step but did't make a change.

Thanks.

Jirka02/16/2007
Any excess water over the infiltration capacity is removed by the surface runoff. The code report both potential fluxes (in your case precipitation) and actual fluxes (in your case infiltration). Difference is the surface runoff. This works similarly for potential and actual evaporation. Check the mass balance error (WatBalR in Mass Balance Information). If this value is below 1-2% then the numerical solution is likely correct.

hCritS value has real meaning only in HYDRUS-1D, which can acctually accumulate water at the surface for later infiltration or evaporation. In 2 and 3D we do not have that capability.

Jirka

## ID = 746, Virus transport – equilibrium vs kinetic sites

jessestimson02/17/2007
I am trying to model with two kinetic sites (AttachSolid1, DetachSolid1, AttachSolid2, DetachSolid2) for virus transport. I am interested in knowing whether I can assign the percentage of Site1 and Site2, or if that is solved for by the model. I noticed that there is a parameter named "Fract." under "Solute Transport Parameters" Window--is that the percentage of Site1, the percentage of equilibrium sites versus kinetic sites, or something else? The manual does not give any discussion in this area that I can find. Thanks for any help that you can offer.

Jirka02/17/2007
Jesse,
The variable "Fract." is used only with the classical two-site model that assume that one fraction of sorption sites is at equilibrium with the solution and the other fraction has kinetic sorption. The colloid sorption model (i.e., attachment/detachment model) does not use any "Fraction" variables. You can view it as if it is the part of the attachment coefficient, since if we use the variable for Fraction of sorption sites, we would just have to multiply the attachment coefficient with this constant. Thus there are two attachment coefficients for different sorption sites (or sorption and straining sites), each independent of the other.

Jirka

## ID = 747, Internal nodes

Erica, 02/18/2007
I read in the manual where you can specify internal nodes for specific conditions, can this be done in the 2D mode? Is there a way to specify internal conditions (i.e. specific pressures) for specific nodes?
Thank you

Jirka, 02/18/2007
Erica,
Yes. When you are in the Boundary module, go to initial conditions. Select any node and click on the Set Value button on the side bar. In the dialog window that will appear, specify the desired pressure head and check the "Internal Pressure Head Sink/Source" check box.
Jirka

**ID = 749, Data output - spreadsheets**

Royce  2/19/2007
I would like to view my model out puts, specificaly theta, in spreadsheet form (rather than graph form). I haven't figured out how to do this. Is this possible?


Jirka    2/19/2007
Each project has its own folder with both input and output data. You should be able to open those with a spreadsheet. The path to the project is shows in the project manager (if you use it). If not, then you should know where the project is saved.
J.

**ID=750, Two site virus modeling question**
jessestimson    02/26/2007

I have a further question about two-site virus modeling. I am have tried to fit one site attachment and detachment to a breakthrough curve, but can see that the two site model will fit better (as suggesting in Schijven and Hassandizadeh).

However when I set katt2, kdet2, katt1 and kdet1 all to 'fit' and run the model, it only fits katt2 and kdet2 with a 95% interval. For katt1 and kdet1, it gives me the same values as for katt2 and kdet2, but with a 95% interval that spans zero, i.e. $1.35 \times 10^{-4}$ min-1 (1.35-10-4 to 1.35 10-4). Is there something else I should be fitting?

I also have a more theoretical question. I do not see still how to fit the relative percentage of Site 1 and Site 2. I try to imagine the set up for the attachment/detachment model and imagine that one independent variable, apart from the attachment and detachment rates for Sites 1 and 2, is the fraction of Site 1 (or the fraction                            of                            Site                            2).

For example, I might be working with quartz (higher detachment rates) and iron oxyhydroxide (lower detachment rates), but I would need to also set the relative fraction of each site, ie with less iron oxyhydroxide, there would be less effect on the breakthrough curve from the lower detachment rate. I wonder where this is represented in the model, or if it is assumed that the ratio is 50:50?

Thanks for your help. Jesse.


**ID=751, Interconnecting**

Brano  02/28/2007
Dear Jirka,
I have question about interconnecting two or more models Hydrus 1D. So I could put them into the space, one soil profile will be communicate with another. How is it possible to do this, if it is? I heard from Dr. van Genuchten that something like this you did in past (may be still).
Thank you.

Brano

Jirka    02/28/2007
Brano,
I'm not sure what you are refering to. We do not have any option for two independent HYDRUS-1D simulations to communicate. The only effort in this direction that I'm aware of is the coupling of HYDRUS-1D with MODFLOW, where multiple H1D simulations (vertical columns) are connected through groundwater evaluated by MODFLOW.
Jirka

## ID=752, Input of experimental psi- theta

Moumita  03/02/2007
Hi,
I have psi(suction)-theta data from Buchner Funnel experiment for a soil. Is it possible to input this experimental data in HYDRUS directly and have the software compute the fitted parameters for psi - theta curve and also compute the unsaturated conductivity K-psi data for that soil?

Jirka    03/02/2007
It is possible. BUt it is much simpler using the RETC program, which is specifically desing for this purpose. You can download it from our web site also.
Jirka

Moumita  03/04/2007
Can you tell me how can I input the data in HYDRUS? I know I can do it in RETC but I would like to do it in HYDRUS directly.
Thanks
Moumita

Jirka    03/05/2007
HYDRUS is a program that simulates flow, not evaluates retention data. If you have some flow related data then you can also include independent retention data. See the example for the onestep outflow experiment that considers one retention data point (the water content at the wilting point).
Jirka

## ID=753, Variable head BC

Mnimmer  03/04/2007
Greetings to all -
I have a few issues that I can't seem to work through. I would add first that the previous postings as well as the examples have been a big help - however, I still can't

seem to get my head around some things. Perhaps others can benefit from my ignorance!

I working with a simple infiltration basin problem. I have field data of precipitation, and head vs. time in the pond as well as in the shallow groundwater table beneath the basin (groundwater is 7' below grade). The soil a very coarse sand/gravel/cobble material (coefficient of uniformity = 125!), so my alpha will be very high. I have a good handle on saturated K from slug/pump tests. I have the grid size set to 3.5 cm and head tolerance at 1 cm. I will eventually run the inverse solution, but currently have iterations set to zero just to compare the observed/predicted results using my initial estiimates of the VG parameters.

The model runs fine if I use a Neumann BC in place of the Dirichlet BC at the ponded surface. However, my flux values are just estimated - the ponded data I have and would like to use. When I enter my observed ponded data (I have 17 hours of ponded data, recorded every 10 minutes - I am running the model for 24 hours total), my result is a very slow run time and resulting heads that are 1E10 meters! I clearly still have a misunderstanding of the use of pressure head as a BC. I had the BC option checked to switch the pressure head to a zero flux when negative values are entered for the time-varying head BC. During times of zero ponding, I entered negative values based on the estimated matric potential (calculated from TDR data and esimates of the expected VG fitting parameters for my soil type).

I understand that a zero pressure head specifies the water table, and that this is corrected by using the special BC option. What I am still not clear on is that once I enter positive values for the pressure head BC, I am back to the same problem of telling the model that the water table exists at the infiltration basin floor BC, when in fact the water table is 7-8 feet below it (and only rises about 1.5 feet during infiltration events).

Sorry for the long explanation, but I know how difficult it can sometimes be to diagnose a problem without proper information. Any help would be most appreciated. Thanks and have a good day.

Regards,
Mike Nimmer, P.E.


Jirka, 3/06/2007
Mike,
Based on what you described, I would set up the boundary conditions as follows:

a) "Time-variable pressure head 1" inside of the pond with "Var.H-1" equal to the ponding depths. I would additionally (in addition to having zero flux when pressures are negative; although you may also try the proceeding option from this dialog with GWL>99999999) also check the option to interpolate (smoothly) head with time.

b) "Time-variable pressure head 2" at the bottom of the domain with "Var.H-2" equal to the height of groundwater level above the bottom of the domain.

c) Atmospheric boundary on the rest of the soil surface with specified precipitation and evaporation values.

I would try this first with less nonlinear soil hydraulic properties, e.g., loam, only once that would work I would use your own properties. It may happen that your properties (extremely high alpha or n) are so nonlinear that the solution does not converge and you need to use finer discretization. But at present you do not know whether this is the reason or the problem is not well defined.

Good luck
Jirka


Mnimmer  04/05/2007
Jirka -
I have a follow-up to my infiltration basin problem. Thanks for the previous comments on the correct use of the variable pressure head boundary - this is working fine. I now suspect that I am adding too much water to my system, possibly due to the improper entry of time-variable BC data. In the time-variable boundary data entry table, an example of how I have added vaules is as follows:

Time (min) Precip (cm/min)
10 -0.05
20 -0.1
30 0.0

I interpret this as having 0.05 cm/min of rain for 10 minutes between the 10 and 20 minute time period; then 0.1 cm/min of rain for 10 minutes between the 20 and 30 minute time period; then no rain starting at 30 minutes.

Is this the correct way of thinking of this? The reason I am not sure is because of an example in Computer Session #5 (Part A) of the manual used for the training session in Golden, CO, last May. That example was simulating flux for one full day, twice per week. The following values were used:

Time (days) Var. Fl1 (cm/day)
1.0 -60
3.5 0
4.5 -60
7.0 0

I understand the flux for day #1, but I am unclear why a line for day #2 is not entered with a flux value of zero, thereby terminating the flux entered for day #1.

Any clarification of this would be most appreciated, and thank you again for the time.
Regards,
Mike


Jirka,   04/05/2007

Mike,
No, it is different. This file

Time (min) Precip (cm/min)
10 -0.05
20 -0.1
30 0.0
is interpreted as having 0.05 cm/min (you can give precipitation values as positive; it does not really matter since the code will assign the right sign itself) of rain for 10 minutes between the 0 (initial time) and 10 minute time period; then 0.1 cm/min of rain for 10 minutes between the 10 and 20 minute time period; then no rain starting at 20 minutes until 30 minutes.

Jirka

## ID=754, Clarification on the use of iPSi

Jennifer  03/06/2007
Hello,
For the Attachment/Detachment modelling, I understand that iPsi can be one of 4 functions. When iPsi=0, does that mean that the dimensionless colloid retention function in equation 3.24 of the manual is 1?
Thanks for you help.
Jennifer

Jirka    03/06/2007
Jennifer,
That would be correct. There are the following two conditions in the program:
if(iPsi1.eq.0.and.SMax1.gt.0.) iPsi1=1
if(iPsi2.eq.0.and.SMax2.gt.0.) iPsi2=1
J.

Jennifer  03/06/2007
So if I want to use clean bed conditions, I need to also have a non-zero value for Smax? Does the value matter?

Jirka    03/06/2007
That's correct.
iPsi=1, Smax=0.
J.
 Jennifer  03/06/2007
Thank you!

## ID=755, Adding solutes inside model domain

Ryanj  03/07/2007

Greetings,

I am interested in predicting contaminant concentrations entering the water table for two solutes (solute one conservative; solute two nonconservative) under varying atmospheric conditions over ~ 200 hundred years. However, the source term I am trying to simulate is located inside my domain (similar but not equivalent to a buried drum).

I would like to incorporate a constant solute source-term concentration (for a handful of nodes) in the middle of my model domain but have not discovered how to implement this in HYDRUS-1D.

Any insight would be greatly appreciated.
--Ryan

Jirka,  03/08/2007
Ryan,
Declare the area where you want to have a solute source as a different material layer. Then specify all parameters the same except for the zero order production. This will lead to the release of solute at a constant rate at this location.
Jirka

Ryanj  03/08/2007
Jirka, thank you for the suggestion. I have a quick follow-up question to ensure that I understand the issue conceptually.
Does adding the zero-order rate constant (for a dissolved phase) amount to the material layer releasing X mg/L per unit time step, and therefore an increased number of nodes in this layer does not amount to more mass being released rather finer spatial discretization?
--Ryan

Jirka03/09/2007
Look at the governing equation. If you specify it for the liquid phase, then it will be gamma*theta*t*length the node represent, i.e., [M/L3/T]*[L3/L3]*[T]*[L]=[M/L2].
J.

## ID=756, Import of GIS data

Luisa   03/08/07
Hello,
I would like to create a model domain that represents a hillslope. I have a set of xyz-coordinates that describe the bedrock topography and another set that describes the overlying soil surface. Soil depth varies. The points have the same xy-coordinates but different z-values. Each data set consists of 230 points on a regular grid (2x2 m).

I know that it is possible to import GIS data (xyz-points) and to create a 3D domain from that. Is it also possible to create these two surfaces or layers within one solid?

Thank you very much, Luisa


Mirek  03/08/2007
Luisa,
Unfortunately this is not possible in the current version. This option was planned for version 1.01 but we have not finished it because of a lot of other work. I think that this option would be very useful and at the same time it is not so difficult to program it. Therefore we will try to do it very soon – I believe that it could be finished within one week. We will have to release new installation 1.02 anyway because one important fix for Windows Vista is missing in the latest version 1.01. I'll let you know as soon as it is ready.

Regards Mirek


Luisa   03/08/2007
Mirek,
thanks for your prompt reply! That'd be great - I am looking forward then to the next Hydrus version.
Luisa


**ID=757, Material properties**

Luisa   03/08/2007
Hello,
I would like to create a model domain that represents a hillslope. I have a set of xyz-coordinates that describe the bedrock topography and another set that describes the overlying soil surface. Soil depth varies. The points have the same xy-coordinates but different z-values. Each data set consists of 230 points on a regular grid (2x2 m).

I know that it is possible to import GIS data (xyz-points) and to create a 3D domain from that. Is it also possible to create these two surfaces or layers within one solid?

Thank you very much, Luisa


Mirek  03/08/2007
Luisa,
Unfortunately this is not possible in the current version. This option was planned for version 1.01 but we have not finished it because of a lot of other work. I think that this option would be very useful and at the same time it is not so difficult to program it. Therefore we will try to do it very soon – I believe that it could be finished within one week. We will have to release new installation 1.02 anyway because one important fix for Windows Vista is missing in the latest version 1.01. I'll let you know as soon as it is ready.

Regards Mirek

Luisa   03/08/2007

Mirek,

thanks for your prompt reply! That'd be great - I am looking forward then to the next Hydrus version.

Luisa

## ID=758, DISC – floating error – missing file?

Jan      03/12/2007

Hey,

A couple of days ago I started using the DISC software for calculating the hydraulic conductivity curve based on tension infiltrometer measurements. At first I had every time a floating point error, but changing the length units form m to mm helped me overcome that problem more or less. But it still isn't possible to commence the simulations because the programm asks for a missing/blank file name, UNIT 70?

I tried various things that I could think of, but none of them worked out, for that reason I hope there's someone here on this forum who has encountered the same problem and hopefully knows a solution for it.

Kind regards,

Jan

Jirka   03/13/2007

After installing the program, have you tried to run examples that came with it. Have they worked or not?

J.

Jan      03/13/2007

Yes, I did the riverside example with is delivered with the programm and that simulation succeeded without any problem.

If I adapt one of the example files with my own data and parameter values, I can simulate my situation but I can't see any difference with the files I created earlier, when I started from an empty disc-file. Working with these latter files always results in an blank/missing file error.

Greets,

Jan

Jirka   03/13/2007

Use finer mesh. It is possible that for your conditions the code does not converge for the course mesh. The error message you get may be result of that. Once there is nonconvergence (or some floating point error), the code (FORTRAN) is unpredictable.

J.

**ID=759, Cumulative solute flux**

Guifang   03/14/2007
Hello,
I'm a beginner for HYDRUS, I use H2D to simulate a field experiment of bromide transport with root uptake. According to the experimental data, the plant uptake is about 50% of the applied bromide mass. I'm searching for a suitable cRoot value for this mass uptake. I have tried many value for cRoot and read the cumulative solute flux(CumChR) in the output data(solute1.out).
My question is what exactly this CumChR signifies? How is it affected by the root distribution and root depth? Can it be directly used as the mass uptaken by root without considering the length unit in [M/L]? When a large cRoot is used, all solute in soil will be uptaken, isn't it? Why there is still solute in my graphic display profile when I use a cRoot and CumChR don't augment anymore?

Jirka    03/14/2007
when croot is large then all solute dissolved in water is taken up together with water. Thus the amount of water is proportional to the water taken up and the dissolved concentration. Since roots can not take up all water (only up to a wilting point), there will obviously be some solute left (as is water) in the profile. This is because HYDRUS simulates only passive uptake, i.e., uptake of solute dissolved in water taken up by roots. I'm working on a conceptual model that will include both passive and active uptake. However, that is not yet included into the publicly available code.
J.

Guifang  03/15/2007
Thank you for your answer.
But I still don't know how I can simulate a 50% uptake of solute?
Guifang

Jirka03/15/2007
There is no such option. The code simulates that roots take up solute up to a certain concentration, not up to a certain fraction of concentration.
J.

Guifang   03/15/2007
thank you for your response
guifang
**ID=760, Mass balance**

Guifang   03/19/2007
Hello,
I have got a very large mass balance error (about 60%) only for a solute nonlinear adsorbed. I have set a very fine discretization close to the soil surface(<1cm). What

may be my problem? Is it possibly a problem with the iteration criteria for nonlinear adsorption? How can I solve it?

Thank you.
Guifang


## ID=761, Soil core air drying in lab

Tleao  03/22/2007
Hi,
I am attempting to model a soil core air drying in Hydrus 2D. I have a 6 cm internal diameter x 6 cm height silt loam soil core that is initially saturated. The core is then exposed to air at both ends for 5 days (the side walls are confined in an aluminum cylinder). I tried to model this using atmospheric boundary conditions in both ends, and time variable boundary conditions. However when I use the prescribed hCrit described in Hydrus book (or higher) the water content did not decrease from saturation. Is there a better way of modeling this? I known the water content should decrease because I have observed data.

Thanks,
Tairone


Jirka 03/22/2007
Try to make at least one row of nodes on each side slightly unsaturated, e.g., -1 cm. Water content will be almost the same and thus the mass balance will be the same as well.
Jirka


tleao 03/23/2007
Thanks Dr. Simunek,
It works very well now. Do you mind telling me where can I get hCrit values for several soil textural classes (e.g. silt loam, sandy loam and clay loam)? Would you have a reference for Table II.1 on page II.4 of Hydrus manual (hCrit for silt, sand and clay) (Rassam et al., 2003)?

Tairone Leao

quote:
Originally posted by Jirka
Try to make at least one row of nodes on each side slightly unsaturated, e.g., -1 cm. Water content will be almost the same and thus the mass balance will be the same as well.
Jirka


Tleao 03/24/07
Just as a follow up to my previous question (please see above)--

What method would be recommended for calculating potential evaporation to input in hydrus evaporation modeling? Is there a recommended equation (e.g. a modified Thorntwaite)? Most examples in the book use potential evaporation = 1. Is there any physical basis for using that value for any soil type core drying at a lab under relatively controled temperature and humidity?

thanks again,
Tairone


Jirka03/24/2007
Tairone,
I usually use -150 m for hCritA. For course soils you may want to increase that since there is no water in the soil at this pressure, while for fine textured soils you may want to decrease that.
I usuallyu use Penman-Montheit or Hargreaves equations to calculate ET.
Jirka


## ID=762, OBS points

Jicerman  03/23/2007
The GUI limits me to 10 observation nodes... is it possible to extract the time series from more than 10 nodes?
Thanks.


Jirka  03/24/2007
In HYDRUS-2D you can indeed have only 10 observation nodes. If you need information from more locations, you would need to rerun the same project with new location of observation nodes.

In the new upgrade (HYDRUS (2D/3D)), you can have up to 30 observation nodes.
Jirka


## ID=763, Geostatistics

TABICHOU  03/26/2007
Can DYDRUS3D generate random fields of Ksat. If so, does it do it 2D or 3D.


Jirka    03/27/07
HYDRUS can generate random values for Ksat, and alpha and theta parameters of the retention curve, but only for two-dimensional applications.
Jirka


Botros 10/17/2007

I can generate random field of saturated hydraulic conductivity (Ks) and van genuchten parameters (alpha, n) outside Hydrus using Geostatistical Software Library (GSLIB). I'd like to implement the spatial variability of the soil hydraulic properties using these values instead of assigning values of (Ks, alpha, n) for a mean water characteristics curve and three sets of scaling factors; alpha(K), alpha (theta), alpha(h). Is there any way to do that?

Thanks,

Farag

Jirka    10/17/2007

Farag,

You need to assign to each node of the transport domain a different material number and then insert the generated soil hydraulic parameters into the appropriate place of the selector.in file. You need to take into account coordinates of each node so that you assign the right material to a right node.

Jirka

Botros 10/17/07

Well, I'm not sure if it's possible to write nodal information in the file selector.in

For simplicity assume that I have only 2 materials (NMat = 2) and the domain contains 1000 nodes. and I'm using van Genuchten's model with six parameters for the soil hydraulic properties model (i.e., iModel=0).

My understanding is that I have to write only two lines at the designated location of the file selector.in (because NMat=2) with only the van Genuchten's six parameters in each line (reference parameters). I can certainly do that also in the water flow parameters module. Then in the designated location of DOMAIN.DAT file, I should have 1000 lines (Nnodes = 1000) where at each line I should write the material index (1 or 2) and three scaling factors (Axz, Bxz, Dxz) that relate the soil hydraulic parameters of this node to the reference parameters.

This is one way to implement spatial heterogeneity of soil hydraulic parameters. I want to implement the spatial heterogeneity in a different way and wonder if this possible in Hydrus. At EACH NODE I want to define different values of the van Genuchten's six parameters and I'm not going to use scaling factors. As I said previously, I can generate these values for each node outside hydrus. Can I implement the heterogeneity that way in hydrus?

Thanks,

Farag

Jirka    10/17/2007

Farag,

The approach is the same if you have only 2 soil materials or 1000. If you have 1000 then you need to write 1000 combinations of soil hydraulic parameters in selector.in and assign corresponding material numbers in Domain.dat. Except that in your case every node will have its own material number and corresponding parametes in the selector.in file.

Jirka


Hirotaka  10/29/2007

I am copying a part of SELECTOR.IN where I have 1250 materials with different VG parameters and Ks. You can create SELECTOR.IN similar to this one. Hope this helps.

```
NMat NLay hTab1 hTabN NAniz
1250 1 0 0
Model Hysteresis
0 0
thr ths Alfa n Ks l
0.098 0.334 5.324 1.76 3.682 0.5
0.102 0.33 5.035 1.874 3.497 0.5
0.106 0.329 5.221 1.925 3.206 0.5
0.102 0.324 5.119 1.917 2.874 0.5
0.099 0.317 5.095 1.907 3.062 0.5
0.103 0.311 4.754 2.056 3.886 0.5
0.103 0.305 4.378 2.229 4.585 0.5
0.104 0.303 4.3 2.291 4.564 0.5
0.099 0.319 4.032 2.209 4.383 0.5
0.084 0.329 3.814 1.969 4.27 0.5
0.087 0.337 3.738 1.956 4.042 0.5
0.095 0.348 3.939 1.949 3.569 0.5
0.096 0.344 4.448 1.816 3.139 0.5
0.093 0.346 5.022 1.778 3.22 0.5
0.082 0.339 5.065 1.784 3.455 0.5
0.086 0.331 5.032 1.858 3.788 0.5
...
...
...
```


**ID=764, Nodal recharge for tracer injection**

Luisa   03/30/2007

Hello,

I am currently trying to simulate the injection of a tracer into a well and its subsequent transport in a 2D slope. I thought I could use the option of nodal recharge for this, but I must admit that the concept of nodal recharge is still not clear to me. What does the value that can be specified for nodal recharge mean (units?)? A flux? Is it possible to switch this internal source off after a certain period of time - so as to simulate the time-limited injection of a tracer? Or are other approaches more suitable to simulate a tracer injection experiment?

I understood from reading the manuals that the concept of nodal recharge is only applied to internal nodes. I noticed, however, when I specify a constant flux BC at the surface that then nodal recharge values are assigned to the boundary nodes although those nodes are not internal.

Thank you very much,
Luisa


onurakay  03/30/2007
I guess you can use a variable boundary pressure head condition for a well. Then you can specify the time duration and the concentration (cBnd1) for that boundary.
Onur


Jirka    04/01/2007
Both constant boundary fluxes and internal recharge fluxes are written into the Q vector that is in the Domain input file. Both have to be time independent. The units are [L3/T] for 3D applications. If you specify constant boundary flux, you specify flux in [L/T] and this value is internally multiplied by the part of the boundary corresponding with a particular boundary node [L2], which thus leads to units of [L3/T] (which is displayed as nodal recharge. These two values, i.e., the boundary flux and the nodal recharge, are thus connected for boundary nodes. For internal nodes, this value needs to be given directly in units of [L3/T] as nodal recharge. Both boundary and internal recharge fluxes are in the Q vector in the Domain.dat file. The value of the concentration of this nodal recharge is given in the Boundary Conditions vector (in the Reaction Parameters for Solute 1 dialog window) as a cWell value. Parameter "Pulse Duration" returns concentration back to zero.
Jirka


Luisa04/03/2007
Jirka,
thanks for your explanations, they made things considerably clearer!
I have run a couple of simulations and found that "pulse duration", specified in the solute transport/general info window, doesn't have any effect on the internal nodal recharge - does it only apply to boundary nodes? I understood from your response that it would also turn the concentrations in the internal nodal fluxes to zero.
Thanks,
Luisa


Jirka    04/03/2007
I have looked at the code. The code turned off concentrations only for boundary nodes in the 2D code and for all nodes in the 3D code. I have to obviously unify this. Are you running 2D?
J.
 Luisa  04/03/ 2007
Hello Jirka,
yes, I am running a 2D simulation in the version 1.02 of Hydrus 2D/3D.
Luisa

**ID=765, Horizontal flow in saturated (freatic) zone**

Quantz 04/06/2007
Hello to all,
i have some problems getting results for my task using Hydrus 2D. I would like at first model a groundwater flow in horizontal direction, thus get vectors of velocity. Domain area size is 36m per 43m and of homogenous material 1 (silt). In the center of this domain is put a concrete block (material 2). Domain area lies within saturated zone. Gradient of groundwater is 1,6â€° and slope of material 1 (whole domain without concrete block) is 0.36. So far my settings were like this:
- Discretization: of whole domain (without block) was set to per 0.5m or 0.2m.
- Boundary conditions: on one side constant pressure head on other seepage face, above various pressure and at bottom seepage face.
- Material distribution: i have selected area for second material as concrete block
- Initial conditions: Firstly on whole material 1 was set at above let's say +5 and bottom +48 and with slope of 0.36. Secondly I've set for concrete block at upper level -10 and at bottom -43.

I would like to ask as well for material properties since parameters for different soil are presumed and given for (i think is like this) unsaturated soil and so for saturated zone should the same. Don't know for example what values have alpha and n, Qa and Qm since principaly infiltration is thought only for concrete. Which hydraulic model should I use, van Genuchten-Mualem or Brooks-Corey?

I hope my pressumption for number of layers for soil profile being 1 is correct.
Time step I suppose is best to fit calculation between discretization and hidraulic conductivity for succesful convergation.

Please help, I look very much forward reading answer and I'm also very much thankful to all for any of yours efforts to reply.

Jirka    04/06/2007
If the concerete is more or less impermeable, which I guess you can assume, then I would not include it into the domain. HYDRUS allows you to have internal holes in the domain.

Your boundary conditions are likely not correct. You can not have a seepage face at the bottom, since seepage face has eithere negative (when unsaturated, unactive) or zero (when active) pressures. You should probably use there no flow. On the right side, I would specify constant head on the right side as well.

J.

Quantz  04/07/2007
Thanks for so quick reply. I have changed what was proposed. Now the program stops some seconds after beggining of numerical solution without any error msg. Does exist a log file of program processes ?
Greet,

Erik


Jirka    04/07/2007
Informaton about the solution progress is in the Run_inf.out time.
J.


**ID=766, Upward flow**

Oranse 04/09/2007
I'm trying to simulate the following situation:
a 6cm column, with -1000cm pressure head (very dry)comes with direct contact to a water reserve from the bottom side (constant pressure head set to zero), and zero flux from the upper side (these are the initial and boundary conditions).
I have the following questions:
1) why does the volume start from a fix number and not zero? I tried changing residual water contact to zero (before it was 0.01) and it didn't change the fix value..
2) what is difference between "Volume" and "sVbot"? As it says in the manual that "sVbot" it the sum of all fluxes that goes through the bottom...shouldn't that in the current situation (zero upper flux) mean that both values are the same? I thought maybe it has to do with the saturated water contact but the difference between these values is not fix to the saturated water contact (but it does stop there).

I'm sorry for the long question...but I'm really puzzled...


Jirka    04/09/2007
It should be possible to change the residual water content to zero. I do not see any reason why not.
Initial condition must have water contents larger than the residual water content, since the pressure head for the water content equal to the residual water content is infinity,

Volume and svBot should be similar if the initial water content is close to zero.
J.


**ID=767, van Genuchten parameters**

Stefan  04/12/2007
Hello,
for an infiltration test I want to create some expectation values with Hydrus 1D. Therefor I need some more van Genuchten parameters than I can find in the Rosettadatabase, for example for peat. Were can I find some more Databases for those Hydraulic Parameters.
Thanks!
Stefan

**ID=768, Snow hydrology**

Mahat  04/18/2007
Hi every body,
Need help to run the Hydus-1D for snow hydrology. What data do I need and how to input these data?
vinod


Jirka    04/18/2007
You actually do not need any data. This is based on a simple assumption that when air temperatures are below zero, precipitation falls in the form of snow and above zero in the form of water (there is actually a 4C interval for this transfer). Once there is snow at the surface it melts using the snow melting constant (e.g., 0.43), which gives how much snow will melt during one day for one degree of C above zero. It is all described in the manual.
Jirka


**ID=769, Freezing and thawing**

eagle-zhao  04/19/2007
Hi Hydrus-users:
I try to model the freezing-thawing behaviour based on hydrus-1D. But it seems numerical unstability of freezing code. Please help me to solve it or give me some suggestions. Thanks!

Eagle-Zhao


**ID=770, Poly disperse particle (10nm-1000nm)**

Sharendr   04/23/2007
Hi
I want to model particle ( iron partilces( 10 nm -1000 nm)) transport in soil media. You asked me to go two site model in stanmod. But can I use that model for poly disperse particle where particle size ranges from 10 nm to 1000 nm.Or only is it for mono disperse particle. What is the best model for breakthrough curves of poly disperse particle?

Hari


Ntoride   04/27/2007
Harendra,
It really depends on the situation. As Jirka mentioned on Sep. 6, 2006, the two-site (attachment/detachment) model would be a candidate to model the particle transport.

Nobuo

Sharendr   05/02/2007
Nobuo,
Ye the problem is , my particle solution has various diameters particles. When I transport that solution, in effluent also I have various diameter of particles. In that case can I use that two site model? I want to know that model is suitable for mono disperse particle solution or poly disperse particle solution?

**ID=771, Confidence limits**

Till     05/04/2007
I inversly modelled van genuchten parameters by pressure heads. The best fit had an $r^2$ of 0.97. But the confidence limits are huge. Most lower limits are below zero.
If I use the upper limit of a single value, the $r^2$ is only 0.5. This indicates that the result is sensitive to the parameter, or not?
Can I trust my results from inveres modelling with this large limits?
Can I trust the confidence intervals?

Jirka    05/04/2007
Read the following article how confidence intervals are calculated:

Simunek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

You can then decide if to trust the results or not.
J.

Till     05/05/2007
It sounds like a clear answer: "It depends ..."
I hope the book helps.
Thanks!

Till     05/05/2007
I just recognized: it is only one chapter. The answer must be easy.
Could you, maybe, send me the article?

Till     05/08/2007
Thanks. The article helped to calculate smaller confidence limits. I identified the parameters that had a large interdependency from the correlation matrix. Then I fitted only pairs with large correlation assuming that all other parameters are known. As far as I understand, the limts tell me how much the paramter(s) may change without incrementing the objective function more than 5%.

??Question:
Why are the limits different for fixation of "known" parameters by upper and lower limtits (upper limit = lower limit) than by fixating them (not marked as variable parameter for the inverse solution)?

quote:
--------------------------------------------------------------------------------
Originally posted by Till
I just recognized: it is only one chapter. The answer must be easy.
Could you, maybe, send me the article?
--------------------------------------------------------------------------------


Jirka 05/08/2007
Program analysis the objective function in the vicinity of the minimum. From the shape of the objective function, it determines correlations between parameters and confidence limits. Thus when the objective function is flat with respect of an optimized parameter, this parameter is not well defined and the confidence interval is brad. When the objective function is steep, then the parameter is well defined and the confidence interval is narrow.
J




**ID=772, Poly disperse (various size) particle is a solution**

Sharendr    05/04/2007
Hi,
My particle solution has various diameters particles. When I transport that solution, in effluent also I have various diameter of particles. In that case can I use that two site model? I want to know that model is suitable for mono disperse particle solution or poly disperse particle solution?




**ID=773, Input parameter of metal transport**

Srilert   05/07/2007
Dear all,
I am not sure for setting the input values of metal transport. Firstly, I conducted the sorption experiments and I got the sorption parameters from fitting the experimental data on Langmuir model. Qmax = 1.52 mg/g and b(binding energy)=19.15 L/mg
for input parameter
I know that Kd multiplies bulk density it produces dimensionless number.
and nu multiplies concentration, it must be produce dimensionless number.
In this case, bulk density = 1.1 g/cm^3
Kd=1.52*19.15*1000 cm^3/g
b=19.25 L/mg
is it correct ?
Could anyone comment for my input parameters ?
Thank in advances

Lert


Jirka     05/07/2007
You need to make sure that you have consistent units both in adsorption isotherm
s=Kd*c/(1+eta*c)
and CDE
ro*s+theta*c

Jirka


**ID=777, The setting of pulse input time**

u0048476   05/15/2007
Dear sir
I have a question about if my total observation time is 1day.
To have a solute pulse duration for 30 mins at time step 0.15 days,
before and after the pulse, there is only water purly for maintain steady state. How should I set the boundary conditon for solute transport?
thanks for providing the information
Boris


Jirka    05/15/2007
Run only solute transport. Make it initially saturated with constant head on sides. Use third-type solute BC on the top and free drainage at the bottom. Set "Pulse DUration" to 30 min.
J.


u0048476   05/16/2007
Dear Dr. Jirka
thanks for your kindly reply. please allow me to ask my question from another way. Suppose in my experiment, I am going to apply two solutes for a same duration but in different time step.
**
initial time: 0 day
final time: 1 day
the first pulse comes at 0.2day for 30mins, with concentration C1
the second pulse comes at 0.6day for 30mins, with concentration C2
under the steady state, how or should I set up the time variable boundary conditions?
Boris


Jirka   05/16/2007
For this you can not use the "Pulse duration" option, but you need to use time-variable boundary conditions. There you can specify anything you want.
J.

**ID=778, Spatial resolution for creating FEM?**

Eric     05/22/2007
Hello,
I am creating a 2D domain that is several meters wide and several meters high. Within the domain I have a few very thin layers (geotextile layers that are ~4 mm thick). There are 3 of these layers on top of each other, with soil over and below the stack of geotextile. I am trying to describe the location of these layers in the domain by using internal lines to delineate the top and bottom of each layer. Ideally, I would like to use several internal lines that are parallel (in the horizontal direction) and stacked above each other with a spacing of approximately .3 mm between the layers in the vertical direction. I know that I may be asking a lot of the model, especially given that the total domain size is on the order of meters, but I would like to create the best representation of the physical system that I can, within reason.

My question regards what the spatial resolution in the preprocessor is when discretizing the domain. For example, if I take one of my internal lines and translate/copy it to create a new line that is .2 mm above the original line, the new line does not appear. (Or perhaps it is placed on top of the original line?) If I shift the duplicate line by .8 mm, the new line does does appear on the screen, suggesting that the resolution is somewhere between .2 and .8 mm. In the latter case I get an error message saying that the new line intersects another line above it, though visual inspection of the entire length of both lines does not reveal any points at which they intersect. I assume that the two "overlapping" lines are within some specified distance of each other that the computer programs believes to be the same location (i.e., they are closer than the resolution used within the program). I have tried adjusting the grid spacing to values as small as 0.1 mm but it did not appear to have an effect on this issue. Does the allowed resolution depend on the grid spacing, or is it dependent on some hardwired value in the program? If my expectations for discretizing the model to such small spatial values are unrealistic, any hints about how to represent the thin geotextile layers would be appreciated.

Thanks,
Eric


Mirek  05/23/2007
Hi Eric,
The problem indeed is in the epsilon defining the model resolution/precision. This epsilon is defined as $Eps = R/10000$ where R is a "domain radius", i.e. radius of the circle/sphere circumscribing the domain. This epsilon is used for example to compare positions of two points and if their distance is less than Eps than the two points are replaced by a single point (which is correct because "overlapping points" are not allowed). Perhaps I should explain reasons why we define the Eps in this way and not for example as $Eps = R/100000$:

1/ I can imagine that too small Eps could cause many problems with incorrect definitions of the model. Point coordinates X,Y are usually displayed with 3-4

decimal places in dialogs and other GUI => it would be easy to make a mistake and create overlapping points, etc.

2/ If your model contains details smaller than 1/10000 of the domain radius then I think that your FE-mesh will be huge. A regular (not-refined) mesh would have 10000 x 10000 = 100 000 000 elements which is quite unrealistic for PCs. I know that you can use mesh refinements and perhaps you will be able to create a mesh with an acceptable number of elements but I'm not sure if such mesh would be good for the numerical solver - this is a question for Jirka.

The Summary: I think that the default value for the Epsilon is defined correctly as it is now. However, I understand that you need a solution. The solution could be that we will allow users to specify their own epsilon (specific for a project). With a smaller epsilon you will be able to define those thin geotextile layers but using of this Eps will be "at your risk", i.e. we can not guarantee that all functions (for example manipulations, auto-detection, etc.) will work correctly. This change will be done in the next version (probably in 1-2 weeks).

We will also check the source code to make sure that this epsilon is used correctly in all functions that make geometrical calculations.

Regards
Mirek


Eric     05/23/2007
Hi Mirek,
Thanks for the detailed answer. The approach you have taken to defining the epsilon is logical and reasonable. Of course, it will be nice if you have a chance to incorporate the added flexibility with epsilon into the next update and I can use it to solve my problem. Or, perhaps the additional option will just provide me with a way to get myself into deeper trouble with HYDRUS.

Thanks again,
Eric


**ID=780, Dependence of water retention curve on void ratio**

Cansand          05/28/2007
Hello
I am doing deformation analysis of unsaturated soil using complicated finite element program. Unfortunately the program is incapable of showing soil water retention curve (WRC) dependence on void ratio or bulk density ; i.e the software considers that WRC is inherent parameter of the soil being deformed. Any one can guide on how unrealistic such assumption is. Also, any guidance to references that considers this assumption is acceptable (as far as the coupled behavior of groundwater motion and deformation is concerned) would be hugely appreciated

**ID=781, How should I input the Freunlich isotherm parameters**

Rajapce     05/28/2007
Hi,
I am currently trying to simulate equilibrium adsorption through a soil media. I have Freundlich isotherm parameters (Kd= adsorption coefficient and nonlinearity parameter n) [Freundlich isotherm
$q= KdC^n$].
I am kind a confused with solute reaction parameters in hydrus 2D.
what does Kd,nu and beta stand for and if I wanna use Freundlich isotherm which parameter corresponding to Kd and n (I explain them before).


Jirka     05/28/2007
Kd stands for Kd,
Beta stands for n,
and nu stands for Langmuir coefficient, zero in your case.
J.


Rajapce   05/28/2007
Hi,
Many thanks for your promt reply.
So nu in langmuir isotherm stands for maximum adsorption capacity?
Raja


Jirka     05/29/2007
Yes, that's correct. See the manual for exact definitions.
 J.


**ID=782, Quantity of precipitation and surface dip**

Folkert Bauer  05/28/2007
Hello,
via boundary condition "atmosperic" I let rain a selected amount on an horizontal surface 5m long. Now I give this surface a dip of let´s say 45° without changing its length nor the amount of rainfall. That´s clear: the more dip I give, the more the length of the projection of the dipped surface to an horizontal plane decreases - thus less rain falls on this 5m surface dipping 45°.

Does Hydrus automatically change the amount of rainfall if one give a dip to a surface without changing its length?

Thanks a lot!
Folkert Bauer

Jirka     05/29/2007
No, Hydrus at present does not do that automatically. I'm working on HYDRUS (2D/3D) update so that this is done automatically. J.


**ID=783, Dependence of water retention curve on void ratio**

Cansand    05/28/07
Hello
I am doing deformation analysis of unsaturated soil using complicated finite element program. Unfortunately the program is incapable of showing soil water retention curve (WRC) dependence on void ratio or bulk density ; i.e the software considers that WRC is inherent parameter of the soil being deformed. Any one can guide on how unrealistic such assumption is. Also, any guidance to references that considers this assumption is acceptable (as far as the coupled behavior of groundwater motion and deformation is concerned) would be hugely appreciated


**ID=784, Flux from variable head BC**

Mnimmer   05/28/2007
Greetings -
In reviewing output files to verify mass balances of my system, I find myself in need of verifying how some fluxes are reported. In my situation, I have a variable head BC to simulate observed ponded water depth reduction with time. If the heads that I input for the BC decrease at a certain rate (for example, 3 inches/hour), but the infiltration rate of the soil is less (0.5 inch/hour), I was expecting the difference to be reported as runoff in the v_mean.out file. However, no runoff was reported in this case. The cumulative flux through the variable BC also does not match the total ponded water depth.

With ET being set to zero, once the ponding has gone away I would expect the cumulative flux through the varaible BC to roughly match the maximum ponded depth. To summarize my question, if the variable heads at the BC decrease at a certain rate, does HYDRUS look at this as a flux that needs to dealt with (by infiltration or runoff)?

A second, related question, involves running the inverse solution with this system. The inverse solution ran fine with 10+ iterations when I had the grid spacing under the basin at 20 cm. When I refined it to 10 cm immediately under the basin (so I could reduce a sedimentation layer at the basin floor from 20 to 10 cm), then the inverse solution bogged down. It still ran fine with the direct solution, and ran fine with 1 iteration with the inverse solution, but gave ***** results with more than one inverse solution iteration. (My initial and minimum time steps were set to 1 second.)

Any thoughts on these two issues would be appreciated at your convenience.

Regards,
Mike Nimmer, P.E.
Biological Systems Engineering

UW-Madison

Onurakay  05/28/2007
Hydrus does not report runoff amount. Any runoff is out of the system. That's why your cumulative flux does not match the ponded water volume.
I can't say smt about the inverse solution. Did you try to change the initial condition and see if that makes a change?
Onur

Jirka    05/29/2007
MIke
1. Runoff is in HYDRUS defined only as the "excess runoff process", i.e., the difference between applied precipitation (atmospheric BC) and infiltration (during ponding). It is not associated with any other BCs.

If you specify variable head BC, HYDRUS does not know why the variable head is changing and does not make any assumption about it. It thus can not assume, for example, that the head decrease is due to infiltration (or evaporation). HYDRUS will just use whatever heads you specify at the boundary and calculate infiltration fluxes based on it and pressure heads inside of the domain. These fluxes are then reported. Thus you can relate them to applied head, and calculate runoff manually.

HYDRUS-1D can simulate decrease of surface head due to the balance of infiltration, precipitation, and evaporation, but only as the atmospheric BC (with surface layer).

2. If direct solution and the first iteration of the inverse solution ran OK, while the second iteration did not, that means that the solution was not stable for the set of soil hydraulic parameters selected for this second iteration. Which are those, you can find in the check.out file. See what parameters were used. You may want to constrain the optimized parameters, since it is possible that unreasonable values were selected by the inverse algorithm. HYDRUS does not check this.

Jirka

**ID=785, On assigning hydraulic parameters**

Cansand   05/29/2007
hello
1-I am using RETC to predict the unsaturated hydraulic conductivity. Under Hydraulic Parameters
I am entering the the data that I have ( do not need to be optimized) including the value of Saturated Hydraulic conductivity (KS=5E-007 ).
After running the problem, I go to output ASCI file but I find that that KS was given Zero value. Why RETC did not consider my entered KS. Are the optimized cofficients valid in this case.

2- Is there any way where I can copy selected coloumns in ASCI and paste them in format understood by EXCEl cells. This I need because I want to display several curves in one plot

Thanks for help

Cansand    05/29/2007
Thanks for Jikra for reply:
Well I enter the value of Ks =5E-007
but I do not know why the program shows KS=0.
Other parameters are fine (shown as they are entered)

Initial values of the coefficients
====================================
No Name Initial value Index
1 ThetaR .0625 0
2 ThetaS .4000 1
3 Alpha .0170 0
4 n 2.6800 1
5 m .6269 0
6 l .5000 0
7 Ks .0000 0

**ID=786, On assigning hydraulic parameters in RETC**

Cansand    05/29/2007
hello
If you are familiar with RETC please consider my quiries:
1-I am using RETC to predict the unsaturated hydraulic conductivity. Under Hydraulic Parameters
I am entering the data that I have ( do not need to be optimized) including the value of Saturated Hydraulic conductivity (KS=5E-007 ).
After running the problem, I go to output ASCI file but I find that that KS was given Zero value. Why RETC did not consider my entered KS. Are the optimized coefficients valid in this case.
2- Is there any way where I can export selected columns in ASCI and to Excel cells. This I need because I want to display several curves in one plot

Thanks for help

Jirka    05/29/2007
First, we have discussion forum for RETC and thus please, next time write questions related to RETC there and not to HYDRUS.
1. I do not see this happening and thus can not comment on that. If I enter Ks, it is considered.
2. Just open the output file directly with Excel. In the "Project Manager" find the path to where the files are stored.

Jirka

## ID=788, Editing soil profile

Henrique  06/04/2007
Although I successfully ran several examples of the Hydrus 1D model, it is not clear to me how to edit a new soil profile (profile depth, soil types etc). I could not do it through the graphical interface either. Does anyone have a clue?
Thanks,
Henrique

Nordbär   06/04/2007
Hi henrique,
first you have to enter your desired soil depth and the number of soil materials of your profile in the section „geometry information". Later you have to define the soil materials by entering hydraulic parameters in the section "soil hydraulic parameters". At the end you can locate the different materials in the graphical editor.
Hope this was helpful,
Norbert

Henrique   06/05/2007
Norbert,
I´ll try that.
Thanks.

## ID=789, Fundamental triangulation failure

Brian   06/04/ 2007
Good Day,
I have created a 2D domain with 5 layers, using points and, subsequently, polylines which are are connected to those points. I then successfully created surfaces for each of the 5 layers (each layer has different hydrologic properties.) I then checked the domain and received the following message:

"No errors found. Domain geometry is defined correctly."

I then attempted to generate the FE mesh. After working for approximately 45 seconds, my PC provided the following error message:

"Internal Error No. 28 during generation of 2D FE-Mesh! Fundamental triangulation failed for Surface No. 6, Edge No. 47. The domain is probably defined incorrectly (duplicated or intersecting boundaries, etc.).

I checked the domain geometry again, and received the following message:

"No errors found. Domain geometry is defined correctly."

I am not sure how to remedy this error. Any help would be greatly appreciated.
Brian


Mirek    06/04/2007
Hi Brian,
Please send me the project (ZIP it if it is large), I'll look at it. What version of HYDRUS do you use now? Version 1.02?
Regards
Mirek


Mirek    06/05/2007
Hi Brian,
There really is an error in mesh generation for your surface No.6, we'll find it and fix it. Now you can get around the problem easily if you refine your mesh. Go to the dialog "Mesh Parameters", tab "Main", group "Targeted FE-Size", uncheck "Automatic" and set a smaller FE size. In your case you can set TS=3m (the automatic size was TS=6.3m). Yesterday I sent you your project with generated FE-mesh so that you could continue in your project (I'm not sure if you have received it). I'll inform you as soon as the error is fixed.
Regards
Mirek


Brian    06/05/2007
Hello Mirek-
Thank you for your help with the FE mesh. After I received your file I found an error that I made in entering one of the points. I edited the point, re-created the surfaces, and re-generated the FE mesh using TS=3.00 m as you suggested; it worked fine. My model is running now.

Cheers,
Brian Davis
Senior Design Engineer
Jacques Whitford NAWE


**ID=790, Mass balance computation**

Henrique   06/05/2007
I ran a modified Hydrus 1D example ("drainage"), and I´ve noticed that the sum of the final "cumulative bottom flux" plus the "root uptake" (outputs)is much more than the "cumulative surface flux" (input). Shouldn´t they match? I´d appreciate any reply.
Jirka   06/05/2007
Check the "mass balance information". If the "WatBalR [%]" is smaller than 1% then the calculations are correct. Then the difference may be a change in storage in the column.

J.


## ID=791, Observation nodes

Onurakay   06/05/2007
I have put 30 observation points in a Hydrus3D project. Although, I can see the observation node file in the permanent project file after the simulation, I can not see these results (pressure head, water content) for these nodes at the Results tab in GUI. Has anyone faced this kind of problem?
Onur


Jirka   06/05/2007
Observation nodes are under "Results-Other Information" tab. It is true, however, that the GUI displays results for up to 20 observation nodes. For the other 10, results are only in the ASCII file Obs_nod.out.
J.


## ID=792, "Trench mesh" and groundwater at hill slope

Kharastarez   06/05/2007
Hi all,

1) I'd like to simulate an infiltration trench. The idea is that the trench is filled up with water at once and infiltrate into the soil till it dry out. Therefore I created a "trench mesh" that contains material which represents the water. The material I defined was sand but I changed 2 parameter, Qs as fully saturated or equal to 1 and a big Ks (1000 cm/d). The boundary condition is no flux. However, it did not work out. I expected some change in water content downwards, but the simulation even showed a dry out. Any suggestion?

2) I'd like to simulate ground water fluctuation at a hill slope. I made a domain that represents a hill slope of 10%. The bottom boundary is parallel to the hill slope. I define the bottom as variable head to show the ground water table fluctuation and the soil surface as atmospheric boundary. However, the simulated ground water table was not parallel to the bottom line but rather as it is parallel in horizontal rectangular case. Any suggestion?

3) If I copy a previous successful project in the project manager and make a new project using some previous data but changed the mesh, some properties and run it afterwards, it always affects the previous one (and vice versa). When the new project is okay, then the previous one crashed and I had to run it again. How could I avoid this problem?

Thank you.
Reza

Eric    06/05/2007

Hello Reza,

I don't have any insights regarding your questions 1 and 3, but may be able to help out with question 2. If I understand your question correctly, you assigned a variable head boundary to the entire lower boundary, right? I have not used that boundary condition for a while, but as I recall, you specify the head values as a function of time. I think that the model may apply that head value you specify to the lowest node of the boundary and then assume that hydrostatic conditions exist along the rest of the boundary to calculate the heads for nodes at other elevations along the boundary. This would result in the flat water table that you describe.

One possible way around this would be to assign separate variable head boundaries (possible in H3D, but not in H2D?) to the left and right boundaries (assuming you have a rectangular domain w/ vertical boundaries on the left and right) and a no flow condition to the bottom boundary. I do not think that Hydrus allows for the assignment of a variable head boundary with a specified slope such as you desire along the bottom. This would actually be a nice feature because then flow could exit the bottom of the system, for example in the case of a precipitation event. It would be useful if a user could specify a given head, which is the same everywhere along the bottom boundary and is equal to the depth below the water table, and also specify the head value as a function of time. Otherwise, using the no flow bottom boundary condition might result in considerable groundwater mounding as a result of precipitation.

Hope this helps. (I have been using H3D lately. It has been a while since I used H2D so I'm sure other users will correct me if I have made any errors.)

Good luck,

Eric


Jirka  06/06/2007

Reza,

1) This should work and I have heard from other users that they did the same. I would need to see the project (input file) to evaluate why this trick does not work for you.

2) This was answered very well by Eric and I have nothing to add. HYDRUS indeed does not allow you to specify time-variable head at a slope.

3) I do not think that I understand this question. If you make a copy of a project using the Project Manager, then the original project (the one being copied) is indeed only copied without it being affected in any way. Thus if you want to rerun it, you should get exactly the same results. When you modify the new project, then obviously anything is possible and it is difficult to judge what might have gone wrong.

Jirka


**ID=793, Validation case 9**

Laurin    06/05/2007

Hi there,
I have some troubles following the calculation of moles of the exchanger Y in the validation case 9 of the HP1 manual. The phreeqc.in -file gives 0.3324 moles of Y for 0.13 kg of water. The bulk density is 1.5 kg/dm^3 and the effective porosity 0.13. That means I have 1.5 kg rock per 0.13 kg water. With a site density of 120 meq/kg rock I would think that the moles of Y have to be 1.5*120/1000.
Also I find different ion concentrations in the input files and in the manual. Boundary concentration for C in the manual is 4.21E-005, whereas it is 2.153E-005 in the Hydrus input.
Thanks for help!


Diederik   06/07/2007
hey,
I checked the input of HP1 with the input of the benchmark model CRUNCH.
For the concentration of C in the inflowing alkaline solution, the concentration used in Table 4.12 is used in CRUNCH (C is in equilibrium with calcite). I have used the wrong value in the HP1 input file.

For the CEC: the bulk density is 2.77g/cm3 and not 1.5g/cm³. However, this value was not given in paragraph 4.3.2.1, but it is the value used to calculate cec in moles/1000cm³. I used this value in CRUNCH, but I can not trace it back in Adler (2001). The value of 1.5 g/cm³ defined in the selector.in file (thus the HYDRUS-input) is not used by HP1 - it is just the default value given by the interface of hydrus.
diederik


Laurin   06/11/2007
Have you tried running this case for transient transport? What is the minimum water content that HP1 can handle? Obviously it won't run when the initial moisture content is 0. When I run the recent version of Hydrus with an initial water content of 0 and a constant head boundary of 0 pressure at the top and free drainage at the bottom the water content rises immediately to about 0.035 in the entire domain. Why is this?


Jirka   06/11/2007
HP1 will not run with zero water content since you need to have some water present to be able to calculate chemical reactions. Similarly, HYDRUS will not run with zero water content, since even when the residual water content is zero, then zero water content represent infinite pressure head. Numerical techniques can not handel infinite numbers. You need to deal only with finite numbers.

Jirka


**ID=794, Upward flow**

Srilert   06/06/2007
Dear all,

I would like to simulate bromide ion from the bottom to the top of saturated column. firstly, I set the water content of the column satulated. then, set the bottom bc is flux =3.6 cm/hr. the top bc is constant pressure = 0. As mentioned above, correct ?
How should I set the boundary and initial condition?

Thanks in advances.
Best regards,
Srilert


Jirka    06/06/2007
Srilert,
If the column is saturated and flow steady-state, then it does not real matter whether the flow is upward or downward (for bromide; it would for colloids where gravity plays additional role). In such case I would simulate that as downward flow, initial condition h=0, boundary conditions - constant head=0, and additionally you need to specify Theta_s=porosity and Ks=flux. I would use third type solute transport BC at the top and zero gradient at the bottom.
J.


Srilert    06/07/2007
Dear Jirka,
If I use HYDRUS2D for simulation, I cannot find the the zero gradient. Is there in the HYDRUS2D, If yes, where ? If not, how should I do for setting up this condition?
Thanks in advance.
Srilert


Jirka    06/07/2007
In HYDRUS-2D use the third-type BC.
J.


**ID=795, Hydrus 2D for simulating constructed wetlands**

theju      06/06/2007
hello,,,

am working on simulating the constructed wetlands using hydrus 2d..can someone help me in knowing what are the important parameters neceesary in order to run the model..at this moment i have read the both the user manuals of hydrus 2d and CW2D manual..but still its confusing..

Hope to get reply soon


**ID=796, Initial parameter estimates for inv method**

Mnimmer    06/08/2007 :  16:56:29

This is a follow-up to my recent posting regarding flux through a variable head BC and the inverse solution method. Thanks to all for the responses. After spending more time on this, I am in need of a bit more clarification with trouble-shooting why the inverse method becomes unstable for certain hydraulic parameters selected. To begin with, I determined that in previous attempts when the inverse solution did not run successfully, it was due to a lower limit I had placed on the alpha parameter. Not placing any limits on the hydraulic parameters allowed the solution to run fine.

However, now I have the same issue with another soil layer in the system. In my infiltration basin, I have a very thin (4") layer of surficial silt loam overlaying a very permeable (60% gravel, 40% coarse sand) layer. Because I have particle size analysis data on the sediment layer, I have thus far just used Rosetta to provide hydraulic parameters for this and kept it fixed. I have used the inverse method to solve for the hydraulic parameters of the sand and gravel layer below. Because the sediment layer is clearly determining the flux into my system, I now wanted to run the inverse solution for the sediment layer along with the sand and gravel layer. I also tried running the inverse solution just for the sediment layer, with the sand and gravel layer fixed. Both scenarios result in the system giving **** solutions at various points in the iteration.

This gets back, I think, to the issue of instability based on the parameters used. However, I am not sure how to find which parameters caused it to be unstable. My questions are: First, I am not sure which iteration the instability occurred in (sometimes it runs for quite a while before failing). Is there a way to determine during which iteration the solution failed? Second, once I know which iteration caused the problem, how do I make a correlation between this and the table used in the check.out file? Basically, I am not sure how to find which set of parameters in the table caused the problem. As far as fixing the problem once it is found, might this be affected by initial conditions, or is changing the initial parameter estimates and providing upper/lower bounds the most effective way to address the issue?

Your continued help is appreciated, and thank you for the time.
Regards,
Mike

Jirka    06/10/2007
Mike,
During inverse calculations, the code at each inverse iteration evaluates several direct runs. It first increases by one percent each optimized parameter (one at a time) so that it can find out in which direction the objective function decreases. It then does a step in that direction and evaluates the objective function again. If the value of the objective value is smaller then its prior value (at previous iteration), this set of parameters is accepted and a new iteration is started.

The parameters for the last direct run are always written into the check.out file and thus from this file you can see which parameters are used during the last run, and if the code crashed, for which parameters it did so. If you have no constraints on optimized parameters, the code can choose even parameters that are not physically

possible, i.e., porosity larger than one or n smaller than 1. Thus it may be needed to constrain these parameters.

I hope that this is useful.
Jirka

**ID=797, Hydrus 2D crashes**

Rajapce    06/08/2007
Hi,
I am trying to simulate flow through a rectangular soil strata (610 cm length and 120 cm height). We have two soils a gravel layer is overlying by clayey soil (hydraulic conductivity 5 cm/day). The inlets are distributed at the bottom of rectangular domain (4 in numbers with constact head 130 cm head). top surface is designed as atmospheric head and and at the side wall at the top it has an openning (seepage face) for outflow of water. All other surface are designed as noflux boundary. I have simulated the scenario for 365 days.
I generate rectangular mesh with 610x120 numbers. But program crashes and showed orthomin error. Then I have tried for general mesh generator with very fine mesh but it does not run at all.
Then I moved to only one soil strata (deleted the gravel layer), but still I could not get any results.

I have run the same scenario with only one inlet at the bottom of side wall and it works fine..
So I am looking for some suggestion so that I can run my simulation
Raja

Jirka    06/08/2007
What is your initial condition?

Rajapce    06/09/2007
initial condition: Saturated soil with initial condition is in pressure head

Jirka    06/10/2007
This should work. Can you zip and send me the h3d file of this project to Jiri.Simunek@ucr.edu?
J.

**ID=798, Inverse solution and dual permeability = [☹]**

Hussat    06/12/2007

I am trying to perform an inverse solution to a flow problem where I have specified one of the dual-porosity hydraulic models but after selecting the dual-porosity model (Durner...) and clicking next I get a program crash. My debugger reports

"Unhandled exception at 0x73dd6bbb in Hydrus3D.exe: 0xC0000005: Access violation reading location 0x0000003c."

This bug can be simulated using the Cone.H3D sample in 2D_Tests folder. Click through the model setup and select "dual-porosity" and then wait for the bang.

I am using Hyrdus V1.02 2D standard edition.
Anybody got a work around for this problem?
Cheers
Paul Hutchinson.


Hussat   06/12/2007
Problem solved. It appears this was a known bug as I downloaded ver1.03 just now & the cone simulation now works OK.


**ID=799, Infiltration in clay soil**

Laurin 06/12/2007
This is probably I common beginners question. A search facility for forum topics would be nice .
Why does Hydrus have problems with infiltration (0 head boundary) into initially dry soil (residual water content or slightly above) with the van Genuchten-Mualem model for clay? If it runs at all, Hydrus produces unacceptable water balance errors in the initial steps.
Thanks for help


Admin06/12/2007
Just a remark: There is a search facility - look at the top-right corner of this page (command "search").
Mirek


 Laurin   06/12/2007
Thanks!


Jirka   06/12/2007
You should always check what your initial pressures are when you specify the initial conditions in terms of the water content. You need to realize that you rarely encounter in nature pressures below -500 m (you need to put soil in the oven to get that), which you can easily get if you specify water contents close to the residula water content.
Also, van Genuchten - Mualem soil hydraulic property model is not particularly good for clays. You should use for fine textured soils the option with the air-entry value of -

2 cm. This was discussed here in the forum many time and thus I will nor repeat it here.

Jirka


**ID=800, Are both inflow and storage the same?**

Rwright   06/12/2007
Is the change in storage = CumAtmBC-CumRoot-CumDrain? If so, this value does not match the inflow value in the mass balance. Also my inflow value is negative. Does this mean that my soil is losing moisture? The value starts off very negative initially and then starts to converge to zero but never reaches zero(after 10 years). If this value remains negative will my soil eventually dry out?

I'm simulating a lysimeter with root uptake. I have clayey silt overlying a loam. My top boundary is atmospheric, side boundaries are no flow and bottom boundary is free drainage with very minimal flux exiting.

Thanks in advance


eagle-zhao   07/05/2007
Dear rwright,
Storage equation is right. But inflow just equals with CumAtmBC, namely the cumulative fluxes of upper boundary. And the negative of inflow means that water comes into soil profile.
Zhaoying
¡¡¡¡


**ID=801, ORTHOMIN terminates**

Hussat   06/13/2007
My simulation does not terminate properly, but sits in a loop on the last time step while printing,

ORTHOMIN terminates - too many iterations, time step reduced.

This goes on until I use Ctr-Break. When I look at the output, for example the ObsNod.txt file, it looks fine, so I am puzzled why the computation did complete. I can prevent the problem by choosing different hydraluic properties, but this must be done manually - the inverse solver cannot deal with this problem. I also cannot find out where the problem is when I have to terminate an inverse problem with Ctr-Break. The output files are not created and there is no Check.out file.

I suspect this is a bug because it always happens at the last time step and the output files look fine.

I am using Ver 1.03.

Any help from the developers would be appreciated.
Thanks
Paul Hutchinson.


Jirka    06/13/2007
ORTHOMIN is an iterative matrix solver (based on conjugate gradient method). This means that the matrix assambled after the discretization can not be solved. It typically means that the problems is not correctly defined, but there can be other problems, such as that the code takes too small time step. I would have to see the project to figure out what went wrong. Is it possible, for example, that your print time, final time, or data point time from the objective function are almost the same, with only a small difference?
J.


Hussat    06/14/2007
Thanks for your comments Jirka.
I solved this problem by changing

Water Content Tolerance = 0.001 and
Pressure head tolerance = 1

to

Water Content Tolerance = 0.0001
Pressure head tolerance = 0.1

I had previoulsy selected (0.001,1) as the tolerance to stop my simulation becoming unstable. The simulation was using the maximum number of iterations and huge pressures were produced. This was a bad idea. I changed the K model to the dual porosity Durner etc K model that better represented the flow problem, but failed to put the tolerance back to (0.0001,0.1), hence the simulation produced the "ORTHOMIN terminates..." output.

Once I put the tolerance back right the model has run perfectly. The simulation fits very well with the field data and I am impressed.

Good work with V1.03. I have only switched to it in the last few days because I needed the Durner model and it looks great.

Paul.


### ID=802, Can you import rainfall data?

Rwright    06/14/ 2007
Hello,

Is it possible to either import historic rainfall data or simulate periods of rainfall.
Thanks in advance
Robert


Jirka 06/18/2007
Sure. Just open your file with Excel or other spreadsheet editor and copy precipitation
data into the column "Prec" of "Time-variable boundary conditions" dialog window.
J.


**ID=803, Floating point error – overflow!**

Vivekgalla   06/15/2007
Helo all,
I am simulating axisymmteical flow in a cylindrical column for 130 days. the column
is comprised with 3 regions, the water flow parameters for the 3 regions are
(thetaR,thet-Sat,alpha(m-1),n,Ksat(m/day),L
region 1 - 0.07,0.41,39.5,2.07,486.34,0.5
region 2 - 0.00729,0.456,15.87,1.701,522.18,0.5
region 3 - 0.00396,0.396,230.41,2.907,54412.2,0.5.

The initial conditions are -50m on the top (the soil is very dry), and i have time
variable boundary conditions.
When i try to run the above model, iam gettting Floating point error - overflow!
Can any one advice me what would be the reason for the error.

thank
vivek galla
Univ of Nevada Reno


Eric   06/16/2007
Hello,
One thing you might try is using a different initial head value. A head of -50 m would
be very dry. I have had problems in the past in which the model was unhappy because
I used too low of a pressure head. You might try taking theta R, adding a small value
to it, and determining what pressure head would correspond to that value for use as
your initial condition (IC). For example, if theta R were .038, you might determine
what head corresponds to 0.381 and try that as your IC. Since you have 3 materials, if
you are assigning the same IC to all of them you may have to play around a bit to find
a single pressure head that gives a reasonable initial water content (i.e., something just
greater than thetaR) for all of them. If you aren't happy about having to use a value
greater than thetaR, you can always start at the higher value and then run the model to
let the system drain before you start applying water at the surface.

Good luck,
Eric

**ID=804, pH of inflowing solution**

Laurin   06/19/2007
Hi there,
I would like to know how exactly the pH of the inflowing solution is calculated in HP1. I assume this is done by charge balancing pH according to the concentration of solution's master species set in the Hydrus interface. I modified the validation case 9 such that it has the hydraulic properties of loamy sand, and set the inflowing boundary to moisture content of 0.4 and the rest of the column has theta 0.1. HP1 overestimates the pH of the inflowing solution as 13.7476 instead of 13.2.
Any ideas?


Diederik    06/19/2007
Hallo,
The master species in the hydrus interface are intering the soil, and phreeqc calculates then the pH by charge balancing. The pH of the inflowing solution is in fact never calculated explicitely.

If you put the concentration of the master species of the inflowing water in phreeqc, and the charge balance the pH, this should give you the pH of the inflowing solution. Can you send me your test problem, so I can have a look to it?

thanks,
diederik


**ID=805, Porous emitter simulation**

hpe650820  06/20/2007
Dear Jirka,
I'm trying to simulate a two material iterations, soil and porous emitter. Emitter is gonna be used for watering plants, it is buried 10 cm depth down from surface. I'm using heterogeneity options because I have two materials, soil and porous emitter. Let me tell you that emitter works with positive pressure head so that time variable seepage face boundary condition was prescribed at the porous material. I think that I can use axisymmetrical vertical flow because emitter has a well defined cylindrical shape. The final time of simulation is 240 minutes.

I don't know if Hydrus 2D can handle this type of problem. If I run Hydrus it will give vectors flowing from the middle of my domain. It is supposed that velocity vectors flow from porous emitter to soil. Could you tell me if Hydrus can do this simulation?

When I use values along cross section option it works just one time. If I want to use it again I need to exit Hydrus 2D and run it. Is it a Hydrus problem or I need to refresh views?

I have run several examples and some of them don't work if I use clay or sand texture soil. is it a problem with the range in the internal interpolation tables?

In the Cum_Q.out file the minus sign in column cumQ3 means the direction of the flow, doesn't it?

Thanks a lot
Dr. Hugo Perea


Jirka      06/25/2007
Hugo,
Sorry for the late response, but I have been traveling in Spain.
Well, Hydrus can certainly handle this. We had quite a few paper published on this topic:

Lazarovitch, N., J. Šimùnek, and U. Shani, System dependent boundary condition for water flow from subsurface source, Soil Sci. Soc. Am. J., 69(1), 46-50, 2005.

Gärdenäs, A., Hopmans, J. W., B. R. Hanson, and J. Šimùnek, Two-dimensional modeling of nitrate leaching for various fertigation scenarios under micro-irrigation, Agric. Water Management, 74, 219-242, 2005.

Hanson, B. R., J. Šimùnek, and J. W. Hopmans, Numerical modeling of urea-ammonium-nitrate fertigation under microirrigation, Agric. Water Management, 86, 102-113, 2006.

I can send you PDFs if you email me.
HYDRUS does not have time variable seepage face BC. Seepage face BC is a no flow BC when the soil is unsaturated and zero pressure when it is saturated. Thus you should use time-variable pressure heads. If you use the new software, i.e., HYDRUS-2D/3D, then you can specify that there is a linear interpolation between specified pressures.

I do not know what problems you have with "Cross Section". I have never encountered this. Can you describe this in more detail?

Typically for sands you need finer spatial discretization since the moisture fronts are much sharper. For clays, I would recommend that you use the soil hydraulic property model with -2 cm air-entry value. This model usually describe clays much better than the classical VG model.

Cum_Q.out – negative values represent inflow in and positive values outflow from the transport domain,
Jirka

 Eric    06/25/2007
Hello,
I can't offer any great solution for your cross section problem, but I wanted to mention that I have also experienced this. As I recall, it seems to occur for me after I have been running the model extensively (e.g., after running numerous models, exporting a variety of cross-sections, or something similar. Perhaps it happens after a run is terminated with control break sometimes?). Unfortunately, I can't remember exactly

what conditions cause it. I have found that closing HYDRUS and restarting it solves the problem and I can then create cross-sections the next time I use HYDRUS.

Jirka, for reference, these comments refer to the older H2D, not to the 2D version of H3D.
-Eric


Jirka    06/26/2007
This happens in the old HYDRUS-2D (not in HYDRUS (2D/3D) where this problem was solved) when you do something with the path, i.e., when you save the project using the command "Save As" or when you import initial condition. Doing that you redirect that path (to a different folder) and the program can not find the path to the definition file for the graph. This should not happen by just displaying one graph and trying to do the second graph.
Jirka


Eric    06/26/2007
Hi Jirka,
I hope it was clear in my previous message that I only encountered this error in the old H2D. The cross-section tool in the new H2D/H3D is very good and I have not had any problems with it. I especially like the way that users can assign a particular cross-section and then watch how it changes with time in the cross-section results window.
-Eric


hpe650820   06/27/2007
Jirka,
A positive pressure head is inside of the emitter so that emitter is saturated during 120 minutes, the total time of simulation was 240 minutes. I used a time-variable pressure head. The seepage face boundary condition was used along the height of the emitter. Free drainage in the bottom and no flux BC in the both sides and top of the rectangular domain.

I have already used -2 cm air entry-value in my clay soil as you suggested, it works. Thanks a lot!!!

My problem with the graphical display results is in the values along section option. when the graph is displayed I do click Export all button. it works one time if I want to use the same values along section option again it will not work so that I need to restart Hydrus 2D again. I did not tell you that I'm using Hydrus 2D old version, Sorry!!!

I'm interested in your papers, I will email to you soon.
thanks
Hugo Perea


**ID=806, Unsaturated transport and surface complexation**

Laurin    06/20/2007
Hi there,
Is there any model that can do unsaturated reactive transport together with surface adsorption according to the diffuse double layer model (Dzombak, Morel) or any other adsorption model (CD Music)?
Thanks again for help!


Diederik 06/20/2007
Hallo,
we are currently working (testing) on a new version of HP1 coupling HYDRUS-1D version 3.0 to phreeqc2.13. In that version, also surface complexation is included (and redox). I am half way testing it.
please contact me if you are interested in it (relatively fast).
diederik


**ID=807, Saline groundwater**

Claude    06/23/2007
Dear all, I would like to simulate the uprise of a saline water table into a unsaturated domain, so I use the atmospheric conditions to set experimental data of the groundwater level variations, and set variable pressure conditions to the lower boundary of the domain, with type-3 conditions. However, all I get is an injection of non saline water water instead of the brine. What am I doing wrong ? Thank you for your help


Jirka    06/23/2007
When you specify the solute transport BC, you are asked for the "pointer to the vector of boundary conditions", with valid value being 1 through 3. Then you need to specify cValue1, or cValue2, or cValue3 in the "Time-variable boundary conditisons" dialog. If these values are zero, then indeed you will have inflow of fresh water.
Jirka


Claude    06/24/2007
Dear Jirka,
thank you very much for your quick answer. I have naturally set the cValue1, then cValue2 and cValue3 (I think I have tested all possibilities) to the concentration of the groundwater. However, I still get an inflow of fresh water into the domain.. Can I send you my input files to check for them? Thank
Claude


Jirka      06/24/2007
Yes. J.


Claude, 06/25/2007

Dear Jirka,
thank you very much for your help. As I edited the 'atmosph.in' file I noticed the three extra columns for time-dependent concentration injection, that I couldn't see in the GUI. I was acutally confused with cBnd values in the 'solute parameters' section. Finally after setting indice 2 to the lower boundary, indice 1 to the upper one and introducing the concentration value in Cvalue2 column in 'atmosph.in' file it works.
Claude


**ID=808, Vary concentration with time**

Srilert    06/24/2007
Dear all,
I would like to ask about initial concentration.
how i can input the initial concentration vary with time in column 1D?
cOULD YOU PLEASE Suggest me?
Looking forward for your answer.
Srilert


Jirka    06/24/2007
You can not change initial concentration with time. That is why it is called initial concentration. It is specified only at the beginning of the simulation.
Jirka


Srilert    06/25/2007
Dear Jirka,
I am sorry. I think i misundersood about this. I would like to ask about boundary condition about solute transport. How could I change concentration with time at the upper booundary condition ? for example
at 1 hr., concentration 10 mg/L
at 2 hr., concentration 20 mg/L
and at 3 hr., concentration 30 mg/L, etc.
How should I do with HYDRUS 1D.
Thank you in advances.
Srilert


Jirka    06/25/2007
You need to select time-variable BC also for water flow, e.g., variable head or flux (even if keeping them constant in time). That will allow you to specify also time-variable BCs for solute transport.
J.

**ID=809, Export data into excel**

Vivekgalla    06/25/2007
Helo all,

can anyone tell how to export the water Boundary fluxes (for eg. seepage face flux data) into excel file.
thank you
vivek


Jirka    06/25/2007
Just go to the folder with input and output data when the project is open in GUI and open a particular input or output file with Excel.
J.


Vivekgalla    06/25/2007
thanks Jirka


**ID=810, Unsaturated steady-state flow column experiments**

Valerie    06/28/2007
Dear all,
I have different questions concerning HYDRUS 1D.

1) I would like to fit breakthrough curves from column experiments performed under unsaturated steady-state flow conditions.
It deals at first with experiment with non reactive tracer.
In the set-up used, the feeding solution is applied continuously to the head of the column, using a sprinkler. Unsaturated conditions are maintained by applying a constant suction at the bottom end of the column.
I choose one observation point at the bottom of the column (fractions of the column outflow are collected and analysed).
I know the water retention curve (therefore the van Genuchten-Mualem parameters) of my soil and the saturated hydraulic conductivity value.
I've put for water flow boundary conditions: constant flux for the upper boundary condition and constant pressure head for the lower boundary condition.
Unfortunately I can't have results expected concerning water flow, in particular concerning initial flow rate or the stability of the flowrate. I don't know where can be the problem in my input conditions.
I tried to take information with examples detailed in the manual (examples 3, 4 and 5), but those with steady-state water flow concern saturated porous media.

2) I don't really understand in which cases it's better to use "concentration boundary condition" or "concentration flux boundary condition".

3) My last question is about the difference between the dispersion coefficient in the liquid phase and dispersivity. Indeed these parameters are in relation with pore velocity v. I don't understand when it's better to fit breakthrough curves with dispersivity or with dispersion coefficient. Examples used sometimes dispersivity value (example 4) or diffusion coefficient with dispersivity 0 (examples 3 and 5).

I thank you very much in advance for your help and advices.

Best regards
Valerie


Jirka 06/28/2007
Valerie,
In your case I would do the following:
a) I would run only water flow. Flux condition at the top and head at the bottom. I would then import (using excel) the final steady state condition (Pressure head) as initial condition for the next simulation.

b) Then I would run solute transport. Certainly use Concentration Flux (Cauchy) BC at the top and zero gradient at the bottom. You may actually disable water flow since you already have the steady state profile (calculated in a and imported).

c) Dispersion coefficient is the sum of the molecular diffusion (*tortuosity) and hydrodynamic dispersion (dispersivity * pore water velocity). You can not experimentally separate them unless you run different experiments at different fluxes. Thus it does not matter which one you fit, whether dispersivity or molecular diffusion, since the results should be the same. I would personaly fit dispersivity since molecular diffusion can be obtain for different ions from physical tables (but I would keep it zero anyway).

Jirka


**ID=811, Model larger than 20km?**

Kardan    06/29/2007
How can I define a model domain larger than 20 km in X or Y direction? Hydrus accepts numbers between -10000 and 10000 only.
Thanks.


Jirka    06/29/2007
Kardan,
HYDRUS indeed limits the size of the domain between -10000 and 10000. The reason for that is that I do not believe that the Richards equation (which is a local equation) can be applied on such an enourmous scale (i.e., km). It could be applied on that scale only if you discretize with very fine grid (i.e., on the scale on which you can assume that Richards equation is valid) and that would lead to millions of FE nodes. I understand that you could have seen in the literature applications of vadose zone models on larger scale, but I personally believe that those applications are wrong (as you can not define Darcy-Buckingham law, for fluxes, over large distances).

We can obviously relax it (or allow different spatial units, km) if you wish us to do that, but I urge you to first reconsider whether you indeed want to solve flow on such a scale using Richards equation.
Jirka

Kardan    06/30/2007
Thank you for clarifying the problem!


**ID=812, Objective function plot**

Alice    06/29/2007
Hello
I am using HYDRUS-1, doing inverse modeling, and I would like to plot the objective function together with bottom concentrations (these are the measured values I used for optimization), in the post-processing. Could you, please, advise how to do that?
Thanks a lot
Alice


Jirka    06/29/2007
Alice,
I'm not sure if I understand exactly what you want to do. The objective function varies as a function of optimized parameters. The dependence of the objective function as a function of optimized parameters is usually shown using the response surface graph (I used them in my Disc infiltrometer and Evaporation method papers). To do that, I'm using a special version of the HYDRUS-1D that varies selected parameters and prints values of the objective function that can be then drawn, for example, using Surfer. I could send you that program once I return from Australia (I'm on the way there now) after July 14. Let me know.
Jirka


Alice    06/30/2007
Hello, Jirka, thanks for reply (enjoy your trip, also!)
I use the term objective function in a different way: it is actually the response that needs to be attained, during optimization (i.e., measured values of a parameter, the breakthrough curve). I used HYDRUS before and I remember that I used this feature. Unfortunately, it was quite a long time since then, and I cannot remember how I did that (although I've tried several 'tricks', it did not work out). I hope it makes sense what am I saying :-)...

Yes, I would like to use your program, if possible.
Thanks,
Alice

**ID=813, Water balance error**

Mnimmer    06/29/2007
Greetings -
I am using HYDRUS 2D to simulate wetting front movement and water table response under an infiltration basin. Model runs have matched observed values well, and all fitted parameters and output looks good with one exception. The water balance error is quite high (50% +) during the one hour period before any precipitation or

ponding is added to the system. Once rainfall and subsequent ponding begins the MB error drops below 0.2% and stays there for the rest of the simulation.

I am curious what might cause this initial high error, because no water is being added to the system, and no water should be leaving during this time. (No water should be leaving since my initial conditions are a flat water table with a no-flow BC at the basin center due to symmetry, and a constant head boundary 100 meters away from the recharge basin. With no water being added to the system during the first hour, there will be no gradient to the water table, and hence there should be no flow out.) The boundary.out file is showing a very low flux (1E-8) out of this constant head boundary during the one hour before precipitation is added.

Any thoughts on this issue would be appreciated.
Regards,
Mike

Jirka    06/29/2007
Mike,
This is beacuse of the way how the mass balance is calculated. For details see the manual. In general, any mass balance error is related to fluxes accross boundaries. Since you have no flux initially, any mass balance error appears large, since it is indeed large as compared to boundary fluxes (which are zero). Once you specify boundary flux, the mass balance error becomes small since the boundary fluxes become meaningful. Look always not only on the relative error, but also on the absolute value. Thus I would not warry about the early mass balance error in your case.

Jirka

## ID=814, CFITIM and CXTFIT

Srilert    07/02/2007
Dear all,
I read the details of CFITIM and CXTFIT , i wondered about these 2 programs. If I want to fit the results from tracer and heavy metals transport in soil column, which one is preferential for my experiment? because I under standand these two can fit the non-equilibrium conditions.
Could any one explain these two program's concept.
Best regards,
Srilert

Jirka    07/02/2007
CXTFIT is much more general than CFITIM. With CXTFIT you can do anything that you can do with CFITIM, plus much more. I.e., additional initial and boundary condition, etc. On the other side CXTFIT is obviously more complicated to work with.
Jirk

## ID=815, van Genuchten parameters with m,n variable

eagle-zhao    07/02/2007
From the RETC code, we can dicide van Genuchten parameters with m,n variable. However, when we input the soil hydraulic paremeters in Hydrus-1D, it seems that no possible to set the m value? Quenstion: How do we set m value in Hydrus? Thanks!

Jirka    07/03/2007
The regular (publicly available) version of HYDRUS-1D uses only the Van Genuchten-Mualem model that limits m=1-1/n. I do have a version of HYDRUS-1D that can handle variable m and n, and can send you this version. However, I can do that only after I return to Riverside after July 16 (I'm currently in Australia). Send me an email after July 16 and I will email you this version.
Jirka

eagle-zhao   07/03/2007
Thanks for your quick reply! Have a good time in Australia.

Zhaoying

## ID= 816, Soil salinity problem

Yasserh   07/02/2007
I working with the hydrus 2D. I am studying the soil salinity and effect of irrigation with low salinity on the decrease of soil salinity in flooding irrigation with ditches (open channel drain). the irrigation period is 3 days every 15 days with 5 cm depthof water every day. I entered the intitial soil salinity distribution and initial water content but the result of the salinity distributuion is as the intial one (no change) something wronge. I want to know what is the correct steps abd the choices for the boundary and initial conditions for water content and solute transport? should i use a constant flux (5cm/day) where should i put the initial salinity of the irrigation water.
looking forward to hearing from anybody and thanks

Jirka    07/13/2007
You can use time variable flux as BC for flow from furrow. You need to assign concentration values in the cValue1 column.
jirka

yasserh   07/13/2007
I already used the time variable boundary as following
time precip Cvalue1
3 0.15 1.9
6 0 0
9 0 0
12 0.15 1.9 .......

The problem is the result of the soil salinity distribution after 360 days is the same as intial condition no change, this means that the program didnt do anything for the salinity distribution

I noticed that during the program was running both of the colums
hAtm and hconst gave always 0
and HDrain gave some results for a short time and NAN for the rest of the program

I dont know what is the mistake?
please can you help me

looking forward to hearing from you¨
best wishes
Dr Yasser Hamed
Sweden


Jirka   07/14/2007
If you get "NAN" that means that there was no convergence in the solution and numbers are out of range. You likely do not have the problem physically correctly defined.
Jirka


**ID=817, CXTFIT and equilibrium sorption**

Srilert   07/03/2007
Dear all,
I am just starting to use the program. So, I am wondering some questions about it. I am grateful if anyone explain clealy my questions.
1.In CXTFIT, does it could estimate non-linear sorption isotherm, i.e. Freundlich or Lanmuir?
2.In non-equilibrium sorption, after I choose non-eqilibrium CDE, how should i do if i want to fit paremeters for physical non-eqilibrium or chemical non-eqilibrium?
3.Is it possible to compare the results from CXTFIT with hydrus 1d or hydrus 2d (in mode of non-equilibrium) ?

Thank you for your assistance
Best regards,
Srilert


Ntoride   07/03/2007
Dear Srilert:
1.In CXTFIT, does it could estimate non-linear sorption isotherm, i.e.
Freundlich or Lanmuir?

No, CXTFIT is based on analytical solutions for the linear CDE in terms of concentration. Analytical solutions can derived only for the linear

CDE. Hence, the solutions are only for the linear sorption.

2.In non-equilibrium sorption, after I choose non-eqilibrium CDE, how should i do if i want to fit parameters for physical non-eqilibrium or chemical non-eqilibrium?

As described in p.15-19 in the manual, chemical and physical nonequilibrium models reduce to the same dimensionless form. Input parameters are listed in Table 6.6 and nondimensional parameters are described in Table 3.1.

3.Is it possible to compare the results from CXTFIT with hydrus 1d or hydrus 2d (in mode of non-equilibrium) ?

Yes, it is possible for the steady-state flow with linear adoption.
Again, CXTFIT is based on analytical solutions for the equilibrium and nonequilibrium CDE, while HYDRUS numerically evaluates CDE.

If any further questions, please do not hesitate to ask me.
Regards,
Nobuo

Srilert    07/03/2007
I am not sure HYDRUS 1D AND 2D have options for nonequilibrium condition ? Is it possible I can fit BTCs results with HYDRUS 1D or 2D for this conditions as well. In addition, I can find sorption parameter of Freudlich or Langmuir also.
Please suggest for this.
Thank you in advance.
Srilert

Jirka 07/04/2007
All HYDRUS models can handel both physical (mobile and immobile water content model) and chemical (two-site sorption model) nonequilibrium, as well as nonlinear sorption.
For mobile-immobile model you need to specify "ThetaIm" (immobile water content) and "Frac" (fraction of sorption sites in contact with mobile water).
For two-site sorption model, "ThetaIm" must be equal to zero and you need to specify "Frac" (fraction of equilibrium sorption sites).
Jirka

**ID=818, Groundwater recharge-evaporation**

Norbert    07/03/2007
Dear all,
I would like to simulate groundwater recharge and evaporation at different grwater depths to create recharge and upward loss curves for using it in Modflow.
I set the following: pot.evap from Penman-Monteith, effective precipitation, soil: sand - 10m, atm. BC with surface layer, constant pressure head BC-zero at groundwater

level-and negative above and positive under the table, grass as root uptake, the root zone is up to 0.5m depth.

My question: is this a correct setting for this aim? And I need recharge and upward loss(evaporation) from different depths, which output can I use(is it the flux in the nod_inf file)? I think upward loss can be calculated somehow (pot.ET-Eff.precip+deep recharge).

Thanks in advance for help,
Best regards,
Norbert


Jirka   07/04/2007
Norbert,
All the boundary fluxes (both potential and actual), as well as root water uptake can be either displayed graphically or are in the output file t_level.out. Fluxes at different depths are indeed only in the Nod_inf.out file.
Jirka


Norbert     07/04/2007
Thanks Jirka!
Norbert


## ID=819, Langmuir isotherm

Elsa     07/04/2007 :  11:27:26
Dear all,
I have one question concerning HYDRUS 1D.
I would like to fit breakthrough curves from column experiments performed under saturated steady-state flow conditions.
I am currently trying to simulate Langmuir non linear and non equilibrium adsorption through a soil media. Whereas, I put Langmuir adsorption parameters (Kd = 0.0137, nu = 1e-8, beta =1) and I have filled iteration criteria (absolute concentration tolerance = 10, relative concentration tolerance = 0.01 and maximum number of iteration = 20), the calculated sorption curve is linear.
Are there some other solute transport parameters or particular boundary conditions to fill in order to obtain a Langmuir isotherm?

Best regards
Elsa and Véronique
 Jirka   07/08/2007
Elsa,
These are all parameters that you need to specify. However, your nu parameter is very small. Unless you have large values of the solution concentrations, you are indeed in the linear part of the sorption isotherm. Try to draw it in, eg., excel.
Jirka

Elsa   07/09/2007
Thank you for your answer. I have indeed large values of concentration (~1e9 bacteria per unit of volume).
The calculated sorption curve I am plotting is Sorb1 (printed for each time step) vs Conc1 printed in NOD_INF.out. file. This curve is linear and doesn't fit with the nonlinear theoretical sorption isotherm.
Best regards
Elsa


Jirka   07/09/2007
Sorb1 is the kinetically sorbed concentration on (f-1) fraction of sorption sites (see description of the two-site sorption model in the manual). This is due to the first-order kinetic process. It should just reflect your adsorption isotherm only at large times when equilibrium is reached, but not at short times (depending on the mass transfer coefficient alpha). The instantaneously sorbed concentration (on the f fraction of sorption sites) (I do not print this since that can be easily calculated from the solution concentration (Conc1)) certainly reflects your sorption isotherm.
Jirka


Elsa    07/11/2007
Ok, thank you for these explanations.
With f = 0 and alpha high enough to reach equilibrium very rapidly, should "Sorb1 vs Conc1" curve reflect the Langmuir theoretical sorption isotherm ?
Elsa


Jirka    07/11/2007
Yes. Use small time step.
J.


**ID=820, Solute transport**

Chiaki   07/05/2007
Dear all,
I am using the HYDRUS2D/3D.I am simulating a pond with time-variable water level.I would like to simulate solute transport with evaporation.

I've used the variable head 1 boundary(with Third-type,cBnd2) at the bottom and the atomospheric boundary(with no solute flux,cBnd1) at the top.I've used the No flux boundary at the both sides.

I have a question. In cBand parameter, I was giving cBand2 as 600mmol. But I ran and got result, at the surface observation point, the concentration was 2e+013 mmol/cm3.What am I doing wrong ?
Thank you for your help,
Chiaki

Jirka    07/09/2007
Are you considering root water uptake. Please note that the wilting point (P3) must be smaller than the minimum pressure head allowed for evaporation (hCritA) (in absolute values). If it is not, that leads to numerical instabilities.
Jirka


Chiaki  07/11/2007
Jirka,
Thank you for the reply.
I am not considering root water uptake,bucause no plants.Then,I inputed evaporation value(from 0 to 6.75 cm/day) and preciptation value(from 0.2 to 0.75 cm/day).Transpiration value is 0 cm/day. The value of hCritA is 100000 cm(for Silt).
Is it right?

Regards,
Chiaki


Jirka    07/11/2007
Evaporation value = 6.75 cm/day. That does not seem reasonable. Divide it by 10 and even then it is still high.
hCritA = 100000 cm seems high as well. I usually use something on the order of 150 m.
J.


Chiaki  07/13/ 2007
Dear Jirka,
Thank you for the reply!. I have a mistake.
Evaporation value are from 0.2 to 0.75 cm/day.
Preciptation value are from 0 to 6.75 cm/day.
I take your advice and I change hCritA value from 100000cm to 150m.
I have a question.
I inputed longitudial dispersivity to 0.5cm and transvers dispersivity to 0.1cm.Is it small? In this condition,I have a result of very big and small concentration.
Can you advise me better value of longitudial dispersivity for silt?
Regards,
Chiaki

Jirka    07/17/2007
Chiaki,
This depends on the size of your domain. General recommendation is to have longitudinal dispersivity equal to about 1/10th of the transpor domain, and the transverse dispersivity about 1/100th of the transport domain.
Jirka


Chiaki    07/17/2007

Dear Jirka,
Thank you for the reply!.
Regards,
Chiaki


**ID=821, Mass balance**

Wemea   07/06/2007
Hello,
Currently I am working with HYDRUS 2D to compare solute transport models. I have constructed models with different discretization and different boundary condition. At the moment I am only using type 1 sites (equilibrium transport).

To explain the differences I have encountered, I'd like to see a complete mass balance. At the moment I am only able to see "mean concentrations", and I rather also identify the concentration of adsorbed solute. is this possible when only using type 1 sites?

Thnx in advance!
Kind regards,
Anne de Weme


Jirka    07/07/2007
Anne,
I print into the output only the solution concentrations when equilibrium sorption is used. The reason is simple: the sorbed concentration can be easily calculated from the solution concentration using the adsorption isotherm,e.g., s=kc. The mass balance reported in Balance.out includes both solution concentrations and sorbed concentrations.
I print separately only nonequilibrium concentrations, since those can not be easily predicted.
Jirka


**ID=822, Is it possible to vary root uptake?**

Rwright    07/11/2007
Hello,
Is it possible to vary root uptake? For example during the four season only a certain percent of the maximum root uptake will take place. Also, I tried looking in the boundary.in file to see the root distribution parameters. However, I did not see them. Where can I find this input?

Thanks
Robert
Ph.D. Candidate
University of Toledo

Jirka    07/11/2007
The root distribution is constant with time in HYDRUS-2D. You can change only transpiration with time. The root distribution function, beta, is in the Domain.in (Domain.dat) file.
J.


## ID=823, Anisotropy

Mnimmer    07/12/2007
Greetings -
I am using HYDRUS 2D and am not clear how to specify anisotropy in hydraulic conductivity. Per the manual and previous submittals, the angle between the principal direction of the K tensor and the x-axis is positive in the clock-wise direction. I would like Kh and Kv to be 90 degrees to one another, so that Kh is in the direction of the x-axis and Kv is in the direction of the z-axis. To achieve this, would it be correct to enter +90 degrees for the angle? Also, I'd like to clarify the ratio of the components. I'd like a 10:1 anisotropy (with the higher K in the x direction). Would it be correct to enter 0.1 for component 1 and 1.0 for component 2?

A final question on an un-related topic. I would like to specify an initial ponding depth in a basin, but then allow HYDRUS to predict how long it will take for ponding to recede based on soil type and other system parameters. I'm guessing this is not possible, since it is a boundary condition and therefore needs to be specified with time.

Any thoughts on these issues would be appreciated, and thanks for the time.
Regards,
Mike


 Jirka   07/12/2007
Mike,
The angle of anisotropy in the 2D code is the angle between the x axis and the first component of the tensor of anisotropy. Thus If you want to have anisotropy only in x and z direction, you should enter zero. Only if the tensor is inclined you would enter nonzero value (e.g., for slope conditions with anisotropy corresponding with the slope).

For 10:1 anisotropy (with the higher K in the x direction), you should enter 10 for the first component (in x direction) and 1 for the second one (in the z direction). These values than multiply Ks, which should be equal to the vertical conductivity.

Jirka


## ID=824, Vertical flow on slope with constant head Bd

Rwright    07/18/2007
Hello,

I'm modeling the vertical flow of a landfill cap section on a 5%(left to right) slope. My boundaries are atmospheric(top), no flow(right side), constant head(left side) and free drainage(bottom). For the constant head boundary I'm using 18% moisture which is the same as the initial condition. When I look at the CumQ.out file I see there is flow in the CumQ1 column which I assume is the constant head boundary since CumQ6 is the bottom flux. Is this horizontal flow or is it actually vertical flow at the face?

Thank You
Robert Wright
Ph.D. Candidate
University of Toledo
Toledo, Ohio

Jirka   07/18/2007
CumQ1 gives cumulative flux through the constant head boundary. Thus it is horizontal flow.
J.

## ID=825, Initially contaminated medium

Xtian   07/19/2007
Dear all,
in a column packed with contaminated sediments I want to simulate the concentration changes in the column with time and space C(x,t) when fresh water is flowing through the sediments. How can I set the initial and boundary conditions that the column is initially homogeneously contaminated?
Thanks

Jirka   07/19/2007
Specify constant initial concentrations throughout the column, zero concentration at the inflow, and zero gradient at the outflow.
J.

## ID=826, How to determine the heat transport parameters

linda_zhhui   07/20/2007
Hello,
I don't know what are the meanings of the heat transport parameters,and I don't know how to determine the values of them,I need your help,thanks!

Jirka   07/20/2007
Linda,

The meaning of these parameters is described in the HYDRUS manual. I will not repeat it here.

You can keep the thermal capacities at default values given in HYDRUS GUI. Similarly you can use values for the thermal conductivities as suggested for different textures in the GUI.

The best way to determine these parameters, if you do not want to use the default values, is likely using experiments with the heat pulse probe. I give here several references:

Bristow, K. L., Measurement of thermal properties and water content of unsaturated sandy soil using dual-probe heat-pulse probes, Agric. and For. Meteorol., 89, 75-84, 1998.

Bristow, K. L., G. J. Kluitenberg, and R. Horton, Measurement of soil thermal properties with a dual-probe heat-pulse technique, Soil Sci. Soc. Am. J., 58, 1288-1294, 1994.

Campbell, G. S., C. Calissendorff, and J. H. Williams, Probe for measuring soil specific heat using a heat-pulse method, Soil Sci. Soc. Am. J., 55, 291-293, 1991.

Saito, H., J. Šimùnek, J. W. Hopmans, and A. Tuli, Numerical evaluation of the heat pulse probe for simultaneous estimation of water fluxes and soil hydraulic and thermal properties, Water Resour. Res., 43, W07408, doi:10.1029/2006WR005320, 2007.

Regards,
Jirka


**ID=827, Calculation error**

Sbromley   07/20/2007
We are having a significant problem when trying to operate our version of Hydrus-2D (version 2.103). We seem to be unable to actually calculate and create output files. This is not only true for new input files that we have been creating, but also for older model input files that have previously run successfully and produced the expected post-processing files. Can you please advise as to what might be causing this problem?


Jirka   07/20/2007
This is such a general question that it is impossible to answer it. You would need to provide many more details. Operating system, where you install it (path), etc.
J.


Sbromley   07/23/2007

Fair enough - I wanted to be sure that this wasn't a common problem with a simple update available before getting into details.

Two of us have Hydrus installed on our local hard drives and have had no problems in the past running Hydrus files on our network or from our hard drives. We use Hydrus in Windows XP Professional for relatively simple applications (i.e. we are hardly researchers and our company has bought Hydrus for a few specific applications).

I have Hydrus in C:\Program Files\UCR\Hydrus-2D (the default during installation). Input files have been saved everywhere from My Documents to our Network drive.

My most recent input files have been creaeted in Hydrus and saved in My Documents. When I run these I get no Post-Processing files. For some older files that have produced many post-processing files previously, only a "Soil Hydraulic Properties" post-porecessing file is produced when I hit run now.

I have contacted our IT support in relation to our Network and operating system and whether any significant changes have been made (I am unaware of any), but considering that I am unable to open old input files and create new ones with Hydrus, I feel it must be more realted to the calculations and settings within Hydrus.

Sorry for the long-winded description - as you can tell, the internal workings of software is not my expertise!

Stephen Bromley


Jirka    07/23/2007
Stephen,
If you execute the computational module and it create the input file Check.out (tht contains summary of input data, as well as soil hydraulic functions) that means that the code starts running well and goes though input files. If it does not generate any other output files, then likely your project is not well defined and the code never start running. Try to save into the same space some of the HYDRUS-2D test examples and run those there. If yuo can do that, then the problem must definitely be in your new projects.
J.


Sbromley    07/24/2007
Thank you Jirka. Your advice worked. When I tried to run one of the test files from the network I got the same problem. The file path was too long due to it being on our network drive. Since I have saved my project into the Hydrus Program File, it is running great.


**ID=828, Deep drainage parameters**

Yasserh   07/22/2007

Please i want to know how I get the deep drainage parameters Aqh and Bqh, there is only one equation at the manual and they are two vaiables


Jirka    07/22/2007
I do not have that information on hand right now. But since I have answered similar question several times in the past in this discussion forum, you should be able to find it here somewhere.
Jirka



## ID=829, Durner K formula

Hussat    07/23/2007
In the Hydrus-3D manual under the heading
/Flow and Transport Processes
/Water Flow Parameters
/Soil Hydraulic Properties

the formula for K in the Durner dual porosity model has formatting problems (on my PC anyway).


Jirka07/23/2007
I do not understand your comment. In the PDF file of the Technical manual?
J.



## ID=830, Time variable boundary condition

Rajapce    07/23/2007
Hi,
I have recently bought a Hydrus 3d standard. In that package I found that in time variable boundary condition there is one window for surface area associated with transpiration. So how one calculate the same? Is it the area (soil surface) covered by plants or the surface area of the plants above the ground.
Raja


Jirka    07/23/2007
Correct.
"Surface area associated with transpiration" represents area (soil surface) covered by plants. The total tranpiration from the entire transport domain is then calculated as "Surface Area" * "Transpiration" and distributed over the entire root zone.
Jirka

**ID=831, Horizontal drainage (Hooghoudt equation)**

Kimi   07/23/2007
Dear All,
I would like to simulate soil water transport with the tile drains using HYDRUS-1D. I chose a horizontal drainage as the bottom boundary condition. I have some questions.
1) How can I calculate the drain discharge rate (q_drain)? How can I decide the watertable height above the drain at the midpoint between the drains (h_dr)? Are these values output to the output files?
2) Which does the calculated soil profile exist at the drain or at the midpoint between the drains?
Best Regards,
Kimi

Jirka    07/23/2007
Kimi,
1) The formulas how the drain flux is calculated are given in the HYDRUS-1D manual. They are calculated from parameters of the drain system, i.e., distance between drains, depth of the impermeable soil layer, depth of GWL at midpoint, etc. The watertable height above the drain at the midpoint between the drains is the result of calculations. It is the location of GW in the soil profile.
2) The soil profile is considered to be at the midpoint between drains.

Jirka

**ID=832, Mesh**

Onurakay   07/23/2007
Is there a way to export an ASCII file that has the coordinates of the mesh nodes? My intention was to create a file that has the pressure heads at each mesh node. I can get the ASCII file for pressure heads sorted according to the mesh node number. How about the coordinates? Thanks.

Jirka    07/23/2007
In the dialog where you choose to convert binary files into ASCII, youcan do that not only for particular variables, but also for nodal coordinates. I believe that it is the MeshTria file.
Jirka

**ID=833, Help**

Matteo   07/24/2007
Dear all,
I am trying to simulate water flow and the transport of energetic chemicals in three soils. After running, I get plots
in the GUI. If I right click, I can see the data. How can I export or copy the output data from Data grid Editor

to Excel for plotting? Is there another file that I can use?

What's Conc (Soil Profil Summary)? And Sconc?
which unit I can use?
Thanks!

Jirka    07/24/2007
All data shown in various graphs have their corresponding ASCII output files in the project folder (see the project manager for the path to them and manual for names).

Conc [M/L3] - solution concentration
SConc - concentration of the nonequilibrium phase (concentration in the immobile zone [M/L3] or kinetically sorbed concentration [M/M])
J.

Matteo    07/24/2007
Thank you Jirka!!!!

**ID=834, Boundary condition for new Hydrus 3D standard**

Rajapce    07/25/2007
Hi,
Before I have worked on Hydrus 2d (old version). Where we have to specify the boundary condition for each edge.
Now I am trying to use Hydrus 3d. I am trying to simulate flow through a rectangular box whose top surface is open to atmosphere.
The other three surface is closed. on left hand vertical surface I have a constand head(or flux) openning. On the right edge I have a drain.
How can I set up those in new version.
The new version I found is way different than the old version

Raja

Jirka    07/25/2007
Raja,
Seting up of boundary conditions is very similar to the old version of HYDRUS-2D. Click at the Boundary Condition Tab at the bottom of the view window. At the right hand you will see an Edit Bar with all boundary conditions that are available. Select the one you want to assign and then select the part of boundary where you want to assign this BC with the mouse.

It is described in detail in the online help under "How to ...
Jirka

**ID=835, Fitting v**

Mjar    07/27/2007
Hi all,
I used CXTFIT to fit my BTC data with the deterministic nonequilibrium CDE to estimate D and V. I checked V as a fitted parameter. But, the program gave only D value. What could be wrong?
Thanks,
mjar


ntoride  07/28/2007
Mjar:
Probably you use dimensionless time T = vt/L. In this case, v is a known parameter. Since it is generally difficult to know exact theta, I would recommend to estimate v, but you need to use dimensional time to estimate v.

If necessary, please send me a project folder + (project name).cxt file. I will have a look at your input parameters.
Nobuo


Mjar    07/31/2007
Hi Nobou,
You are absolutely right! When I used the dimensional time, the problem was solved. Thank you so much for your help.
May I have another question?

The length of my column is 10 cm and I would like to output for only one position (at the end of the column). I set the characteristics length to be 10 and input 1 for the initial value of output position. But, one of my friends suggested that the characteristics length should be 1 and the initial value of the output position should be 10. Is this true?

Thanks again,
mjar


ntoride   07/31/2007
mjar:
You can use an arbitrary value for L as long as the same value is used for parameters listed in Table 3.1. If you use 10 cm, for example, you will have an omega value which is ten times larger than that for 1 cm. This input parameter is originally designed for multiple BTCs with dimensionless time and positions.

Nobuo


Mjar    08/01/2007
Dear Nobuo,

Thank you very much for your prompt response. I checked Table 3.1 and I think I understand what you meant. As you can tell that I am very new to CXTFIT and am trying to analyze my data while I am studying the CXTFIT manual.

Would you please kindly take a look at one of my projects? I think that I have made mistakes somewhere but I cannot figure out what was wrong. My fitted V and D values seem too samll. I am trying to estimate D and V from conservatice tracer BTCs using both equilibrium and nonequilibrium CDE. I use both methods to compare the r^2 obtained from the two methods and to check if there is a physical nonequilibrium in the small (6.12 cm D X 10 cm L) undisturbed soil column. I used CFITM or CFITLM to estimate R.

How may I send you my project folder and .cxt file?
I greatly appreciate your time and thoughtfulness.
Thank you,
mjar


Ntoride     08/06/2007
Majar,
Sorry for my late reply. Please zip your project folder and .cxt file, and mail it to ntoride"at"bio.mie-u.ac.jp. (replace at with @)
Nobuo


Mjar     08/07/2007
Hi Nobuo,
I understand that it takes a lot of time to help everyone who has posted questions on this forum. I have just zipped my project folder and .cxt file and mailed the file to you. Thank you very much for all your helps and I look forward to hearing from you.
Thank you,
mjar


mjar     08/13/2007
Hi Nobuo,
I am wondering if you receive my project folder that I sent to you via my aol email account. I am afraid that the email may be filtered out because the attached is a zip file.
Thank you,
Jarai


**ID=836, CXTFIT**

mb7982   07/30/2007
hi.I don't know any thing about application time , help me plz .


ntoride     07/31/2007

I am willing to help you, however, please carefully read the CXTFIT manual and work on some examples close to your problem to make your question more specific.
Nobuo


**ID=837, Variable flux BC**

Luisa    08/02/2007
Hi,
I am trying to model the effect of interception on subsurface flow on a forested hillslope in 3D. I am using precipitation data of a rainstorm event as atmospheric boundary condition along the hillslope surface. I have two scenarios: 1) only atmospheric boundary condition along the entire hillslope surface (i.e. spatially uniform infiltration); 2) I have been trying to simulate the spatially variable input that is created by the vegetation by using a time-dependent variable flux boundary condition at some spots along the hillslope surface, assuming that throughfall is 95% of the precipitation at these spots.
The simulation for scenario 1 runs through without any problems, giving plausible results and no indication of surface runoff. The simulation of scenario 2 terminates at a certain time step, showing a sudden strong increase to absurdly high values in boundary fluxes and pressure heads along boundaries. When I checked boundary fluxes and pressure heads in the output files of the two scenarios, however, I found no differences until this time step; the evolution of boundary fluxes and pressure heads are almost exactly the same. Decreasing the minimum time step helped in some cases.
My questions are now:
- What could be the reason for this behavior? Is the variable flux BC not suitable as a surface boundary condition?
- What else in the output files could I check to get a clearer idea about what went wrong?
- Is there another way to realize spatially non-uniform infiltration in Hydrus?

Any thoughts and suggestions would be appreciated!
Thank you, Luisa


eagle-zhao    08/02/2007
Hi,Luisa,
Since you consider the throughfall from the precipitation, have you reduced this part from the potential evaporation? In additional, how do you estimate for throughfall, is it the daily value or not?
Zhaoying


Luisa    08/03/2007
Hi Zhaoying,
At the moment I don't consider evaporation or transpiration at all, I am rather focusing on the subsurface flow and trying to find a way to realize in the model a spatially non-uniform input. As for the estimation of throughfall: I am working with 15 min data (hope this answers your question).
Luisa

**ID=838, Output problems**

Quantz   08/05/2007
Hello,
hydrus 2d encountered problem with writing output after some hours of process. I think my hard disk space doesn't limit output.
Best regards!


**ID=839, On bulk density [g/cm^3] used in RETC**

Yam     08/06/2007
Is the BD[gr/cm^3] used in the Neural network procedure of RETC:

a- The saturated unit weight bulk density: (Gama) or
b- The unit mass bulk density: Gama/g (where g is the gravity accerlation) or
c- The dry unit weight bulk density:: GamaD= Gama-n *GamaW
(where n is the soil porosity and GamaW=the water unit weight density
or
d- The dry unit mass density :GamaD/g (where g is the gravity accerlation)

If it is a or c then the dimension should be (gf/cm^3)
Thanks


**ID=840, Inverse solution**

Srilert     08/07/2007
Dear all,
I just have one question about inverse solution in CXTFIT and HYDRUS 2D. when I perform column experiment but I cannot get BTCs because heavy metal have a lot of sorption capacity. However, I can know the varied depth concentration in column by sectioning column extraction. Therefore, I know concentration in each depth. How can I get the retardtion factor from these 2 models? I am not sure I saw in "Data structure Input code", it shows Z(I), C(I) for a fixed time. Can I use this option for input my data for estimating sorption parameters (retardation factor) ? In addtion, I don't know in case of HYDRUS 2D?
Could anyone please suggest to solve this problem?
Thank you very much.
Srilert


Ntoride   08/09/2007
Srilert
Yes, Z(I), C(I) for a fixed time is designed for concentration vs. depth. In order to estimate R, it is necessary to know v in advance. If you know exact theta, v can be estimated from water flux. We generally use nonreactive tracer to determine v based on a BTC or concentration profile. Please also note that you need to choose resident

concentration c_r or total resident concentration c_t = Rc_r depending on your measurement.
Nobuo


Srilert    08/10/2007
Dear Sir,
Can CXTFIT used for unsaturated zone like HYDRUS 1D or 2D ?
Thank you
Srilert


Ntoride    08/10/2007
Srilert
CXTFIT is based on analytical solutions for the equilibrium and nonequlibrium CDEs and HYDRUS uses numerical techniques to solve Richards eq. and CDE. I would strongly recommend you to read CXTFIT and HYDRUS manuals.
Nobuo


## ID=841, Inverse solution

Srilert   08/07/2007
Dear all,
I just have one question about inverse solution in CXTFIT and HYDRUS 2D. when I perform column experiment but I cannot get BTCs because heavy metal have a lot of sorption capacity. However, I can know the varied depth concentration in column by sectioning column extraction. Therefore, I know concentration in each depth. How can I get the retardtion factor from these 2 models? I am not sure I saw in "Data structure Input code", it shows Z(I), C(I) for a fixed time. Can I use this option for input my data for estimating sorption parameters (retardation factor) ? In addtion, I don't know in case of HYDRUS 2D?

Could anyone please suggest to solve this problem?
Thank you very much.
Srilert


## ID=842, Very long execution time

Rajapce    08/09/2007
Hi,
I am working on Hydrus 3d standard. I am trying to simulate a solute flow through a rectangular box (610 cm length, 122 cm height and width).[Domain definition: hexahedral].
Initial condition: saturated with water

bottom 20 cm gravel layer and rest is silt loam with low hydraulic conductivity (5 cm/day).

Solute was bromide. Simulation period: 730 days

Boundary condition:three side of the box have no flow boundary. Top surface is set to atmospheric boundary condition.

At left hand wall there are three constant head node. At the right hand side has (top surface) two seepage face node.

I have tried to simulate the above scenario and found that run time was too long.. After oneday, the program has only completed 19 days simulation.

I worked on Hydrus 2d before. The model was very fast and has taken only few minutes to complete the simulation. So I am wondering whether there is other reason for that slow execution time.

Raja


Mirek    08/10/2007

Hi,

Calculation in the new version (HYDRUS 2D/3D) is not slower than calculation in the old version (Hydrus-2D & Meshgen). You can verify this if you import the old project to new HYDRUS and recalculate results. There can be more various reasons why your calculation is slow, may be that Jirka will know immediately where the problem is (wait for his comment). If your problems continue then send us your project, we'll look at it (hydrus@pc-progress.cz).

Mirek


Jirka    08/10/2007

I think that the calculations should be faster in the new version since we are using faster compilers. If you indeed have a project that ran faster in the old version than the new one, please send it to us (both old and new projects) and we will look at it and try to figure out why it is.

Jirka


Sparks    10/25/2007

Hi Jirka

I have just started using the 3D part of the new software and am experiencing the same time issues. I have not inputted my whole problem, ie not all soil types and boudnary conditions. And it is taking about 4 hours to simulate for one year - any assistance or ideas?

Thanks

Jacqui


Jirka    10/25/2007

Jacqui,

How big is your problem? How many nodes you have? What are your boundary conditions (daily fluxes)? How nonlinear are your soils (n=?)? I would need more information to be able to comment.
Jirka


**ID=843, Inverse solution**

Srilert    08/10/2007
Dear all,
I just have one question about inverse solution in CXTFIT and HYDRUS 1D. when I perform column experiment but I cannot get BTCs because heavy metal have a lot of sorption capacity. However, I can know the varied depth concentration in column by sectioning column extraction. Therefore, I know concentration in each depth. How can I get the retardtion factor from these 2 models? I am not sure I saw in "Data structure Input code", it shows Z(I), C(I) for a fixed time. Can I use this option for input my data for estimating sorption parameters (retardation factor) ? In addtion, I don't know in case of HYDRUS 1D? because I saw many options for inverse solution for solute transport

Could anyone please suggest to solve this problem?
Thank you very much.
Srilert


**ID=844, How can I do the registration?**

Ahmed    08/14/2007
hello
please i want to know how can i do the registration for hydrus 2d
and i need to know how can i used this programe
its the first time i use it


Jirka    08/14/2007
Ahmed,
To be able to use the HYDRUS software package, you need to purchase a licence from one of the HYDRUS distributors. You can get their contact information on http://www.pc-progress.cz/Pg_Hydrus_3D_Resellers.htm

Best regards,
Jirka


**ID=845, Retrieving nodal values of one mesh layer**

Luisa   08/15/2007
Hello,

I am running simulations of water flow in an irregular 3D domain and I am interested in the distribution and evolution of pressure heads in a particular mesh layer and how these values change under different scenarios. I understand that the nodal values of pressure heads for all time steps are reported in the file "h.out" that is used for the graphical output and the flow animation. How is this file organized? E.g. the first value is for node no.1, the second value for node no. 2 and so on? Is it possible to retrieve the numbers (numbering) of the nodes that are contained in one particular mesh layer?

Thank you very much,
Luisa


Jirka     08/15/2007
Louisa,
It is indeed organized as you wrote, ie., sequentially based on node numbers. As far as I know there is no simple way of exporting values for only a part of the domain.
J.


Mirek   08/15/2007 :  11:12:30
Yes, it is as Jirka wrote. Just several remarks:

1/ Use command "Convert Output to ASCII..." (in the main menu->Results) to convert h.out to text file (H.TXT). Then you will see there pressure heads for each time layer. These values are written sequentially, 10 values are on each single line and then it continues on a new line. The order corresponds to order of FE-Mesh nodes.

2/ If you are interested in values at a specific node(s) then you will need to display numberig of FE-Mesh nodes. You can switch it on at Navigator -> tab "View" -> Numbering -> FE-Mesh -> Nodes (all check boxes must be checked to see the numbering)

3/ I think that numbering of FE-mesh nodes is increasing with increasing z-coordinate and therefore FE-mesh nodes in a standard horizontal layer can be represented by "from-to" indexes. For example: from "Nmin" to "Nmax" where Nmax-Nmin = number of nodes in one horizontal FE-mesh layer. This number should be same for all standard mesh layers unless you modify the mesh by command "remove selected elements".

Regards
Mirek


Luisa    08/15/2007
Jirka and Mirek,
thank you very much for your quick and helpful reply!
Luisa

**ID=847, Transport with Langmuir adsorption**

Raintree     08/15/2007
Hi,
I just want to set up a run for solute transport with a langmuir adsorption.
I set parameters: beta=1, nu, kd=, frac=1

however, no matter how I change kd and nu,I always get c/co=1, nothing adsorbed on the solid phase.
any one can give me some suggestions on what's wrong with my simulations.

thanks!
Raintree

Jirka     08/16/2007
Check the examples distributed with the software. One of them (I believe that it is Test5 or Test6) has a case with a nonlinear sorption.
J.

**ID=848, Root water uptake**

Dush    08/16/2007
Hi,
What is the purpose of the "Root" column in the soil profile summary? Is this the root zone depth?
I have been trying different depths (assuming "Root" is root depth) with the root water uptake paratemers. However the depth I use (1m, 2m, etc)shows no effect on my modeled results.

Thanks for any insights,
DJ

Jirka     08/16/2007
This is a dimensionless number that is proportional to the root distribution as a function of depth.
Jirka

stathi02us    09/22/2007
If we have a profile of 100cm and we want a uniform root zone of 100cm(depth) then my "Root" value is 100?
thank you very much
Stathis

Jirka    09/22/2007

Stathis,
It does not matter what values you specify (just make it one everywhere), as long as thery are every where the same (uniform distribution). The code internally normalizes these values so that integral over the entire root zone is equal to one.
Jirka


## ID=849, Inverse solution

Srilert    08/17/2007
Dear all,
I found obtion in inverse solution "
Residual concentration, flux concentration, total concentration (liquid and solid)" what do they mean ?
Best regards,
Srilert


## ID=850, Add sorption isotherm models

Srilert    08/17/2007
Dear all,
I would like to know if I have new sorption isotherm like modified Langmuir model, Can i add it into transport model (Hydrus 1d) ? How I can add it ?

Looking forward for your answer.
Regards,
Srilert


## ID=851, Simulation for paddy soil

Lea      08/19/2007
Dear all,
I try to simulate the transport of Cu for a paddy field. Surface layer of water is about 20cm.
1) Which boundary conditions could be suitable for that? Can I use variable Pressure Head/Flux for upper BC and Seepage face for lower BC?
2) And could you please tell me how I can specify 20cm-layer of water above soil profile in Hydrus 1D?

Thank you very much
Lea


Jirka    08/20/2007
Lea,
If the water level in the field is constant, then just use constant head BC and specify 20 cm in the top node as the initial condition.

The bottom BC depends on what is happening at the bottom. It will be either free drainage, if water is allowed to freely drain at the bottom or constant or variable head if there is water table.

Jirka


Lea      09/12/2007
quote:
If the water level in the field is constant, then just use constant head BC and specify 20 cm in the top node as the initial condition.

Hi Jirka,
I do not know we can use Constant head BC as a initial condition or not (If it is possible, could you please tell me how to specify 20 cm of above water?)

I have learnt that "Atmospheric boundary condition with surface layer" is to permit water to build up on the surface. The height [h(L,t)] of the surface water layer increases due to precipitation (irrigation), and reduces because of infiltration and evaporation.

Could you tell me which BC should be selected for the simulation?
Thank you very much
Lea


Jirka      09/12/2007
Lea,
As I wrote before, if the water level in the field is constant, then just use "Constant head BC" and specify 20 cm in the top node as the initial condition.
If you have time variable water level that depends on how much water infiltrate into the profile, then use "Atmospheric boundary condition with surface layer". Again specify the initial water level as positive initial head for the top node and then in "Prec" column add any additions as time proceeds.

Jirka


 Lea      09/24/2007
quote:
As I wrote before, if the water level in the field is constant, then just use "Constant head BC" and specify 20 cm in the top node as the initial condition.

Hi Jirka,
I am very stupid. Could you show me how to specify 20 cm in the top node as the initial condition?
If I simulate a solute transport for a 1 meter-profile of a paddy soil with 20 cm of water above, the depth of soil profile (In geometry information box) should be 1.2 m, is it right? And for material distribution in graphical editor, 20 cm above will be

specified as a layer of water? Then the problem is how to specify "water flow parameter" (Qr, Qs, Alpha,...) for that layer?
Thank you very much


Jirka    09/25/2007
If the water layer standing at the paddy field is constant then you should use constant head BC. Then
a) the pressure head initial condition in the top node must be equal to the specified pressure (e.g., 20 cm) and
b) the transport domain is only the soil profile, i.e., 1.0 m.
If, on the other side, you want to simulate how water from the water layer infiltrates into the soil profile (i.e., water layer depth decreases), then you need to use the "atmospheric BC with the surface layer" (this is only in HYDRUS-1D) and again
c) the pressure head initial condition in the top node must be equal to the specified pressure (e.g., 20 cm) and
d) the transport domain is only the soil profile, i.e., 1.0 m.

Jirka


Lea     10/04/2007
You did a great job!
Thank you very much
Lea


Lea     11/13/2007
Dear Jirka,
I have simulated for a paddy soil domain with 4 layers:
0-25 cm: Clay loam,
25-50 cm: Silty clay loam,
50-75 cm: Loam
75-100cm: Loam
Upper boundary was a constant head pressure. From the simulation for Hydraulic conductivity I got a "strange trend", HC decrease at the bottom of layer 25-50 cm (picture below).

Could you please explain me?
Thank alot
Lea


Jirka    11/13/2007
Lea,
This is the correct solution. It follows from the fact that different layers have different saturated conductivity.
0-25 cm: Clay loam, Ks=6.2 cm/d
25-50 cm: Silty clay loam, Ks=1.68 cm/d
50-75 cm: Loam, Ks = 25 cm/d

At steady state, you reach constant flux (2.76 cm/d) through the soil profile. TO get this constant flux the pressure gradient needs to be different in different layers (Darcy's law). Thus the pressure increases (positive) in the first layer and then decreases in the second layer. Loam can trasfer this flux while being unsaturated and thus the pressure at the top of the loam layer becomes negative. As there is a large pressure gradient at the interface between the two layer, the conductivity is much lower. You should check also the pressure head profiles.

Jirka


## ID=852, Variable bottom pressure head boundary

Dush   08/20/2007
Hi all,
I am modeling infiltration in an 8m soil (sand) column with an atmospheric BC with surf layer on top and a variable pressure head (water table) at the bottom. The water table remain above the bottom boundary (depth to water<8m) throughout the simulation period. The initial conditions are in the water content.

1. How do I set the GWL in the "variable boundary conditions" input as my variable pressure head boundary? is it the depth to water from surface (depth=0) (i.e 6.0 if the water table is 6m below the surface? or 2.0 as above the bottom depth (8-6)?

I have tried both this ways, but the model does not converge. However, the model converges when I use the above numbers as negative values, but the results I think are obviously incorrect.

Thank you very much for any help.
Dush.


Dush   08/21/2007
I figured out how to do this and problem with the convergence. Pressure head has to be placed as the (soil column depth-depth to water). Then increase the default pressure head tolerance somewhat.

Please correct me if this is incorrect.
Thanks


Jirka   08/21/2007
When the depth of GW is 6 meters and you simulate 8-m deep soil column, then the bottom BC is equal to 2 m (8-6).
J.


## ID=853, Inclined base surface

Luisa   08/22/2007
Hello,
is it possible to create an irregular two-sublayer solid (based on GIS data) with an inclined base surface? I have a geometry text file with the first two columns being the x and y coordinates. The third and fourth column both contain the coordinates of the inclined base surface. The last two columns have the z coordinates of the two sublayers (as explained in the user manual). The first import of the geometry works fine, but the second import to create the sublayers does not work, the sublayers that are being created resemble the base surface and not the irregular sublayer surfaces. I haven't been able to figure out why this happens.

Thanks, Luisa


 Mirek   08/22/2007
Hi Luisa,
I believe that this should be possible. Send me the text file with coordinates (mireks@pc-progress.cz), I'll look at it.
Regards Mirek


Mirek    08/23/2007
Hi Luisa,
Thank you for the file. The problem is that points defining your inclined base surface do not lie exactly in a plane and you would need to define their z-coordinates more precisely (i.e. with more decimal places). When you create the base surface (i.e. after the first import) these z-coordinates are fixed to lie in the plane and everything seems to be OK. But the second import overwrites fixed coordinates and the solid is not defined correctly.

Recommended Solution:
1/ Look at pages 104 and 105 of the User manual and read about "Thickness Offsets". The thickness offset allows you having an inclined solid although its base surface is not inclined (i.e. it lies in the XY-plane). We will use thickness offsets for your domain.
2/ Change all z-coordinates in the third column to z=0 and import geometry as before. The domain will be created correctly because there will be no problem with precision of z-coordinates.

Another solution:
Define z-coordinates more precisely so that all points defining the base surface lie in a plane with tolerance < epsilon where the epsilon is defined (visible) in the Geometry Information dialog. You must have the latest version 1.03 to be able to see this epsilon.

Regards
Mirek
 Luisa   08/24/2007
Hi Mirek,
both solutions fixed the problem!

Thank you very much once again, Luisa

## ID=854, Multiple simulations

Dush    08/23/2007
Hi,
I need to do multiple runs of a model with different model parameters (soil/root water uptake etc.) and evaluate the effect of the above on the modeled results. Instead of having to change the parameters manually each time for the different runs, I would like automate the task (i.e. have the parameters for the different model runs assembled in a separate file , make HYDRUS-1D read from that file, run the model, and save the output to a different set of files or an array. Has anyone already done this?, and if so can I get some help?

Thanks in advance,
Dush.

Nordbär   09/06/2007
I had nearly the same problem. I also wanted to automate the procedure of parameter changing. Later I saw that there was an easier way:
I created a new folder for every separate model specification. Then I made a copy of the file 'selector.in' in every folder and changed different parameters in each file in a text editor. After that, I created a batch-file in my model directory with the following content:

Copy Path1\selector.In selector.In
H1D_Calc<return.txt
Copy Obs_Node.OUT Path1\Obs_Node.OUT

Copy Path2\selector.In selector.In
H1D_Calc<return.txt
Copy Obs_Node.OUT Path2\Obs_Node.OUT

...

The first row overwrites the model specification file selector.IN with the varied files. The second row starts HYDRUS (the file 'return.txt' contained a single 'return', so that there was no need to press the return key after each run).
The third line copied the output file 'Obs_node.OUT' into the respective folders.

Later I wrote a little program to manipulate the file 'SELECTOR.In' automatically, but this way was not much faster then the way described above.

I hope this helps,
Norbert.

 Lai    09/13/2007

Hi, I did as Norbert suggested and created a new folder in root directory in D: disc, and then made a copy of all the H1D Modle files and created a batch-file in this new folder. but when I run the batch-file,a hint comes in the MS-Dos Window as "Folder with input data of the specified project does not exist or pathway is too long or corrupted return.txt\Selector.in".
where the problem is? can I get some help?
Thanks in advance.
Lai


Dush   09/20/2007
Norbert,
Thanks for the tip. I have been doing other stuff for a while, but will give your method a try in a couple of weeks.
Thanks again,
Dush.


**ID=855, Change pH in two metals transport in column**

Srilert   08/30/2007
Dear Sir,
I ever use HYDRUS 2D but I know that hydrus cannot perform if pH change. I just want to know HP1 or PHREEQC can solve my problem.
My column experiment:
1.Manganese(Mn) sorbed onto soil in column pH 5, THEN
2.Flush with lead(Pb) into column
3.when influent concentration=effluent concentration, change pH from 5 to 4

Anyone advice me this problems.
best regards,
Srilert


Diederik     08/30/2007
Sir,
from the concise description of your experiment, I think it is possible to simulate this with phreeqc (if mainly steady-state flow conditions prevail, and the average solute transport velocity is rather constant through the soil/column) or HP1. For HP1, it depends which kind of surface complexation model you want to used to model the adsorption.

Most probable you can start in phreeqc (to set up the chemical part of the problem), and then, if needed go to HP1 (version 2.0 which will be released in the near future),

greetings,
diederik

**ID=856, Inner point boundary condition**

Kardan   09/03/2007
How can I define a point as an inner boundary condition of a 2d model? I inserted a point inside a rectangular domain, created a surface that includes this point, but the point is not selectable when defining boundary conditions. I am obviously missing something... Any help is greatly appreiated!
Jaromir


Jirka    09/03/2007
Jaromir,
One can not specify boundary conditions at internal points. Boundary conditions can indeed be specified only at boundaries. You can only declare the internal point to be "Internal Pressure Head Sink/Source", i.e., you can specify the pressure head that is kept constant during the simulation. The node can then act as a source or sink. This is done in "Initial Condition". Select node and click on "Set Value". You will get a "water flow initial condition" dialog where you can check this option (i.e., "Internal Pressure Head Sink/Source").
Jirka


Vaglan  09/12/2007
Dear Jirka,
I would like to simulate ring infiltrometers with Hydrus 3-D.My problem is that i can not specify the wall of the infiltrometer in the domain (no flux condition).
Is there any way to simulate ring infiltrometers with hydrus 3D?
I only tried 2D domain with axisymmentric geometry.
Sorry for my English, is not my native language.
Thank you in advance
Vaggelis Pollalis


Jirka    09/12/2007
Vaggelis,
You are talking about the ring walls incerted into the soil, right? Thas is easy to account for. If that is the case then keep it a part of the profile, but specify both x and z components of the anisotropy tensor equal to zero to elements representing the ring (in the subsurface). USe rectangular type geometry since then you can easily control the thickness of the ring wall. Then water will not be able to flow through these finite elements and will have to seep around it.
Jirka


Vaglan   09/14/2007
Dear Jirka,
if i put a second material with Ks=0 ans i set it as the wall of the ring infiltrometer in the appropriate nodes, is it the same thing?
thanks
Vaggelis

Jirka    09/14/2007

Vaggelis,
No, you can not have zero conductivity. You could use some really small number, but the best way is what I suggested before, i.e., to use the anisotropy tensor. THis is also partly due to the fact that material properties (e.g., conductivity) are assigned to nodes (and thus there would be a linear change over the element from zero to Ks of the other material), while anisotropy tensor is assigned to elements, and thus its domain is defined exactly.
Jirka


## ID=857, Simulation period using H1D

Lai      09/11/2007
Hi,all
I'm a newer HYDRUS 1D user, and recently I am using the H1D to simulate water distribution along a soil profile under natural condition, so the atmosperic boundary condition with surface layer was used. Now my question is that is it possible to run the H1D for three or more years? If it can do this, it seems difficult to input all the daily records of three years' meteorology data.
Does anyone have a clue? Thank you in advance.


Jirka      09/11/2007
Lai,
Yes, it is possible to run HYDRUS for a very long time period. We have run it for hundreds of years with daily input values. You can prepare your input in any spreadsheet, e.g. Excel, and simply copy (Ctrl+V) into HYDRUS GUI. If you have more than 32,000 data records, you can modify directly the Atmosph.in file.
Good luck,
Jirka


Kheatwole      09/12/2007
32,000 records? I just downloaded the latest Hydrus-1D release and it seems to not like anything over 10,000 records. I also had problems with Hydrus overwriting my Atmosp.in file by truncating it to 10,000 lines after I ran a simulation.


Jirka      09/12/2007
GUI will tell you how many records it can handle. There is a warning for that.
You need to modify the atmosph.in file while HYDRUS-1D is not running. You also must not forget to change the "Number of records" in that file (I believe it is on the third line). If HYDRUS is running, then the file is in its memory and then you may overwrite your modifications once you click on save.
Jirka


Lai      09/13/2007
Thank you, Jirka. Now the H1D model can run properly.

But a new problem came up, when I was simulating water distribution for a short period(1 year) by using long period(10 years) meteorology data, all the extra data had to be deleted? because a dialogue box pop-up with "The time for the last time-dependent record must be equal to the maximum calculation time."
Is there any way to switch the simulation period easily with the same meteorology data file? Thank you in advance.

Jirka    09/13/2007
The GUI checks that the last record is equal to the final time. YOu would need to run h1d_calc.exe (computatonal module) outside of GUI to avoid this check.
J.

Lai     09/13/2007
Thank you, Jirka. Sorry for asking you so many questions one after another.
When I ran the model with Root Growth Data for many years(e.g.3 years), it seems only one year's Root growth parameters can be set, and only one year's information of "root water uptake" occurs in the result file. So, how to set the Root growth parameters for a long period.
Thank you very much.

Jirka    09/13/2007
Root growth is indeed implemented only for a single season. J.

**ID=858, Problem with post-processing**

kheatwole    09/12/2007
Hi,
I am simulating only water movement in a soil column with precipiation and evaporation. After the simulation ends I can use all of the post-processing features except for the profile feature. I get an error saying "Error reading .....NOD_INF.OUT file."

Jirka    09/12/2007
Look at the file. Is it possible that some values overflew the format and are shown as stars (i.e., ********)? If that is the case, use different length units so that this does not happen.
J.

Kheatwole    09/12/2007
Yes, that is the case. My profile is too deep (-22250 cm) for that file to handle.
If I switch units to meters will I still be able to maintain the accuracy of my daily precip and evap values without having too many charcacters?

Jirka     09/12/2007
I would think so. J.


## ID=859, Capillary barrier

asakcelik     09/18/2007
Is it possible to simulate capillary barriers using HYDRUS 1D? Thank you.


Jirka     09/18/2007
Yes, why not? Except that 1D can not account for the horizontal flow, which is why usually capillary barriers are built (to divert flow sideways). For that you need the 2D code.
Jriak


Basakcelik     09/18/2007
Thank you for your reply. I'll consider this.


## ID=860, Concentration flux BC in Hydrus 1D

dinhviet122     09/19/2007
Dear all,
I am trying to simulate a solute transport in a packed column packed with downward liquid flow. I would like to understand how to implement the "Concentration flux BC" in Hydrus 1D. I have therefore two questions :

1. I would like to know if the equations, including initial and boundary conditions, implemented within the code are dimensional or dimensionless.

2. What is the variable and its unity that we provide to the code when the boundary condtion "Concentration flux BC" is selected ? The question is asked because there are two unknown variables in the aforementioned BC, i.e. q0 and c0, and only one variable is required and we do not know which one and which unity is used.

Many thanks in adavance and looking forward to hearing from you very soon.
Kind regards


 Jirka     09/19/2007
1. You can express concentrantrations in both dimensionless form, as well in any units you want (mol/L, mg, ppm, etc.). The governing equation is unit independent.
2. You need to specify c0. q0 is already specified somewhere as BC for water flow, or is the solution of the water flow problem. q0 will be internally multiplied by c0 to give solute flux BC.
Jirka

**ID=861, Injection Well**

Lstorer   09/19/2007
Can you give me some input on the best way to model an injection well? I am trying to model different well geometries (vertical vs. horizontal vs. angled wells) and injection rates. I have tried different things but they all seem to have their own problems:

Nodal Recharge - When putting nodes along a vertical line, the model does not converge well. I can put one node with a certian inflow and it will work fine, but when I add another below this point, the water seems to flow around it - I have a pic of this where you can see the issue, it is a little difficult to explain.

Putting a boundary "hole" in the model for the well - When I start to use horizontal and angled wells, the water "pools" in places that it should not.

Using internal initial conditions - I can not easily control the volume of inflow this way.

If you have any suggestions on which method is typically used for this type of application, I would really appreciate it.


Moumita   10/11/2007
I had modelled injection well using the option axisymmetrical vertical flow considering the line of symmetry to pass through the well. Then I had defined constant flux boundary along the sides of the well. I didnot have any problems.
Moumita


**ID=862, Import chart data**

Castanheira   09/21/2007
Hello. I have a litle question about the Hydrus2d.
How can i import the chart data to excel. i.e, the head or concentration for one observation node?

Thanks in advance.
Castanheira


Jirka   09/21/2007
You can open directly with Excell the Obs_node.out file from a folder of a particular object. This file contains information that is displayed in the OBservation Nodes chart.
J.


Castanheira   09/21/2007

Dear Dr. Jirka thanks you for your help,
If i want export to excel the cumulative boundary fluxes. I can i do that? And the seepage cumulative flux?
Best Regards
Castanheira

Sorry, one more thing... If i have only infiltration the cumulative solute flux at atmospheric boundary is negative. True?


Jirka     09/22/2007
Cumulative fluxes are in CumQ.out file. J.


## ID=863, Inverse solution

Srilert   09/24/2007
Dear all,
I got some problems in inverse solution. I would like to explain my scenerio.
1.Firstly, my column was soaked uniform metal concentration.
2.Flush upwardly with DI water and took samples periodicly.
3.I got data of concentration v.s. time and some metal concentration still sorbed in the column (I know concentration with depth at the final time)
4.How can I use the inverse solution of HYDRUS 2D ?

Could anyone help to response this problems?
Looking forward for your answer.
Srilert


Jirka     09/25/2007
Srilert
First, it seems that your problem is one dimensional and thus it would be simpler to use HYDRUS-1D. Second, I guess that your question is not how to simulate such system, which seems trivial (since you likely have saturated, steady-state flow), but on how to define the objective function that should be minimized. You have two types of data and since you can use in HYDRUS only one concentration mode (resident, flux, or total concentration) to define the objective function they should be entered as follows. You can enter them using two "Types". One (Type=4) allows you to enter breakthrough data, and another (Type=14 or 15) allows you to enter concentrations versus depth. You need to use either Type=14 if you have equilibrium (instantaneous sorption) in the system, or Type=15 if you have nonequilibrium (kinetic sorption) in the system. If Type=14 then you have to specify liquid concentration (M/L3), if Type=15 you have to give sorbed concentration (M/M). In both cases (Type=14 or 15), "Position" represent the "Print Time" level. Here is an example:
a) Breakthrough data
Time Conc (liquid) Type Position(Obs. Node #) Weight
1440 0.234 4 1 1
b) Metal distribution with depth
Depth (negative) Conc. (liquid) Type Position(Print time level) Weight

-1.3 0.408 14 2 1
or
Depth (negative) Conc. (sorbed) Type Position(Print time level) Weight
-1.3 0.408 15 2 1

Jirka


Srilert   09/25/2007
Dear Jirka,
I would like to know about definition of "resident, flux, and total concnetration"
AND in inverse solution you means I should one system "type 4" or "type 14 in case of equlibrium condition" in HYDRUS 1D

Another question, I try to simulate inverse solution in HYDRUS 2D BUT it was not fit properly. Could you please suggest this problem for me.
If you can, I will send you my file.

Looking forward for your answer.
Srilert


Jirka    09/27/2007
Srilert,
Type=4 is for concentrations in observation nodes (Pos=1, 2, ...NObs). X=Time, y=concentration.

Type=14 is for concentration at specified print times (Pos=1,2, nPrint). X=depth (negative), y=concentration

For definition of Resident and flux concentration, see CXTFIT manual or "van Genuchten, M. Th., and Parker, J. C., Boundary conditions for displacement experiments through short laboratory soil columns, Soil Sci. Soc. Am. J., 48(4), 703 708, 1984."

Total concentration includes both solute mass in the liquid phase and in the solid phase (sorbed).

HYDRUS-2D always consider only resident concentrations (in the liquid phase). It does not have options for flux or total concentrations. That option was added only to HYDRUS (2D/3D).

Jirka

**ID=864, Solute transport, soil layers**

stathi02us    09/25/2007

Hi, I run a simulation with a layered soil profile. In the first 10cm I have a fine soil (ks=5.7cm/day). From 10cm to 100cm i have a coarse soil (ks=103.54cm/day). I want to simulate a solute transport with the following characteristics. The upper layer has:
Longitudinal dispersion: 0.59cm
Freundlich parameters kd=59.3ml/mgr, b=0.736
For the second layer:
Longitudinal dispersion: 0.64cm
Freundlich parameters kd=14.4ml/mgr b=0.782
The initial condition is that the first layer is contaminated with a concentration 0.1mgr/cm3.(at t=0)
Then I run a 10 year simulation with fresh water.(meteorological data)
The "problem" is that when the solute reach the second layer then I have an increase in a maximum concentration of 0.11mgr/cm3. Has this a physical meaning? Or I have to check the problem again?
The balance error for water flow and solute transport are very good (below 1%)
Thank you very much
Sorry for my English is not my native language.
Stathis


stathi02us    09/25/2007
please visit the following link to view the image
[URL=http://img403.imageshack.us/my.php?image=chartsf6.png]
[/URL]


Jirka    09/27/2007
Stathis,
Do you consider root water uptake? If yes, then it is to be expected that solutions will get more concentrated.
Jirka


stathi02us 10/05/2007
Thank you very much, its the root water uptake.


**ID=865, Two time-dependent Dirichlet conditions**

claude3    09/26/2007
Hello,
I want to simulate the water flow and solute transport in a rain-fed paddy field, where the height of the superficial water level depends on rainfalls but also on the water management of the farmers. On the other side the groundwater level is also time dependent, depending of the dynamics in the recharge area. Consequently I have to deal with two time dependent Dirichlet conditions. How can I solve this with Hydrus2D ?

thank you for your help and advices
Claude


Jirka    09/27/2007
Claude,
Unfortunately, HYDRUS-2D allowed only one time-dependent pressure head (Dirichlet) boundary condition and one time-dependent water flux (Neuman) boundary condition. You will need to keep one of your pressure head boundary conditions constant, or divide simulation into multiple runs and adjust BC between runs.

We have realized that this was a limitation of HYDRUS-2D and therefore we have implemented into the new version of the code, called HYDRUS (2D/3D), up to four time-dependent Dirichlet and four time-dependent Neuman boundary conditions. You would, however, have to upgrade to this new version, if you want to use this option,

Best regards,
Jirka


claude3    09/27/2007
Dear Jirka,
Thank you very much for your answer. I will hopefully upgrade soon to the new version.
Claude


Jtourneb    09/27/2007
Hello Claude
I did a coupling between 2 models (HYDRUS and PCPF) during a collaboration with Japanese colleagues.
The reference of the paper is:
The development of a coupled model (PCPF-SWMS) to simulate water flow and pollutant transport in Japanese paddy fields
Paddy Water and Environment
vol 4(1) 2006
Tournebize, Watanabe, Takagi, Nishimura
Abstract
A new coupled model (PCPF–SWMS) was developed for simulating fate and behavior of pollutant in paddy water and paddy soil. The model coupled the PCPF-1, a lumped model simulating pesticide concentrations in paddy water and 1 cm-surface sediment compartment, and the SWMS-2D, a finite element numerical model solving Richard's and advection-dispersion equations for solute transport in soil compartment. The coupling involved improvements on interactions of the water flow and the concentration the pollutant of at the soil interface between both compartments. The monitoring data collected from experimental plots in Tsukuba, Japan in 1998 and 1999 were used to parameterise and calibrate hydraulic functioning, hydrodynamic and hydrodispersive parameters of the paddy soil. The analysis on the hydraulic functioning of paddy soil revealed that the hard pan layer was the key factor

controlling percolation rate and tracer transport. Matric potential and tracer monitoring highlighted the evolution of saturated hydraulic conductivity (K S) of hard pan layer during the crop season. K S slightly decreased after puddling by clay clogging and strongly increased after mid term drainage by drying cracks. The model was able to calculate residential time in every soil layers. Residential time of tracer in top saturated layers was evaluated to be less than 40 days. It took 60 days to reach the unsaturated layers below hardpan layer.

In case do not hesitate to contact me.
Sincerely
Julien

**ID=866, Inverse solution in heat transport using Hydrus**

Pancho    09/26/2007
Hi,
I was wondering if Hydrus (1D, 2D or 3D) has inverse solution to solve for the parameters of Chung and Horton when modeling heat transfer in the soil.
Thank you very much
Francisco

Jirka    09/27/2007
Francisco,
HYDRUS-2D did not have an option to optimize thermal properties. That option was added only to the new HYDRUS (2D/3D). HYDRUS-1D does have that option as well.
Jirka

**ID=867, DISC**

Erfan   10/02/2007
Hi there,
I have been stuck with the DISC package to inversely solve my infiltration data and estimate the VGM (van Genucten-Mualem) model parameters. I have tried different approaches in order to run the code properly. However, my efforts were all unsuccessful.

The example files were working well in my machine. However for some unknown reasons, my own data causes problems for the code. Particularly, things started to mess up as I define the cumulative infiltrated water.

I hope to get some good feedbacks in order to solve my problem.
Thanks,
Erfan

Jirka    10/02/2007

Erfan,

Check what the differences between your case and example tests are. make sure that you are using the same units everywhere. The code should work as it was used not only by our group, but also elsewhere by other groups, e.g.:

Ramos T. B., M. C. Goncalves, J. C. Martins, et al., Estimation of soil hydraulic properties from numerical inversion of tension disk infiltrometer data, Vadose Zone Journal, 5(2), 684-696, 2006.

Jirka


Erfan    11/05/2007

Hi Dr. Simunek,

Thanks for replying to my former post regarding DISC operation. I finally solved my problem and was able to run the code successfully.

I have another question regarding the code. For all of my simulation runs, the relative error of the first step of infiltration tests is high. The pressure steps that I apply on the field were -25 cm, -15cm, -6 cm, -3cm and 0cm of water respectively. The fitted curve for cumulative infiltrated water has high relative error for the first step which is -25cm.

Is there any way in the code that I can define different weight for each pressure step?

Thanks,

Erfan


Jirka    11/05/2007

Every data point that you enter into the objective function (In HYDRUS (2D/3D)) can have a different weight. I do not remember if it was possible to assign different weights also in the Disc Program.

Greetings from New Orleans (SSSA meeting).

Jirka


**ID=868, Solute units**

Castanheira    10/02/2007

Dear Hydrus2d users

Im simulating a solute transport in a subsurface drainage system, and I have some questions. My unit lengths are in cm, and time are days. I have the concentration input is in mg/l (irrigation water). So I need put the concentration input in mg/cm3 in the variable boundary conditions. i.e: 1mg/l = 0,001 mg/cm3.

However the output concentration is in mg/cm3, but cm3 of what? Soil, soil solution or extract saturation?

Thanks in advance.

Regards

Paulo Castanheira

Jirka    10/02/2007
Paulo,
Concentration units in HYDRUS are always Mass per volume of liquid solution, e.g., mg/cm3.
Jirka


## ID=689, FEM – FDM difference in 1D?

Nordbär    10/03/2007
Hello everybody,
I have a simple question:
I want to compare different 1D soil models. One of them is Hydrus 1D. The other Models are based on the method of finite differences. In general I know the difference between FEM and FDM, but what about one-dimensional models?
Is there a completely different approach for these solutions or are there no discrepancies in 1D-Problems?

Thanks for help,
Best regards, norbert.


Jirka    10/05/2007
Norbert,
In 1D both approaches are quite similar. Actually the Finite Elements method with the mass lumping (which is used in HYDRUS-1D, and which means that the diagonal is made more dominant) leads to the same formulation as the Finite Differences method (that is why we explain in the manual the FD method). Thus the difference is mainly in the implementation of boundary fluxes.

Jirka


## ID=870, Root water uptake

Botros    10/04/2007
I have a question regarding assigning value of beta(n)to the root zone nodes. Should the summation of all Beta(n) nodes equal to 1.0 as suggested in Eq. 2.11 or we actually assign values of b_dash(n) according to Eq. 2.10 and not values of beta(n) itself. If the latter is true, I assume it doesn't matter if we multiply all values of beta(n) by a constant value. Am I right?

I also wonder if there is any way to convert the file DOMAIN.IN to DOMAIN.DAT
Thanks,
Farag

Jirka    10/04/2007
The code automatically normalizes the beta function. Thus the absolute values do not matter, only relative values are important. Domain.in can be saved as Domain.dat using a command on the file menu of the Bounday/Domain module.
Jirka

## ID=872, Linux and Vista

Johnnyzee    10/05/2007
Will there be a Linux version of Hydrus in the future?
We have been using Ubuntu Linux and like it. On the other hand our company is resisting the move toward Windows Vista. If inevitable is Hydrus 2D/3D compatible with Vista?

Thanks,
John Zupancic

Mirek    10/06/2007
HYDRUS is compatible with Windows Vista, although there are several minor issues (see http://www.pc-progress.cz/Fr_Services_Hydrus_FAQ.htm, FAQ 34). We will fix all these problems in the next HYDRUS installation 1.04 which will be released soon – probably in a couple of weeks. However, we still can not recommend Windows Vista as the best OS for HYDRUS. The main reason is that most of 3D graphical accelerators still do not have good drivers for Windows Vista and therefore 3D graphics can be significantly slower than on WindowsXP. But I believe that the situation will be getting better and all important manufacturers of 3D accelerators will be able to fix (or optimize) their drivers within next 6 months.

Linux: unfortunately we do not have sufficient capacity to prepare HYDRUS for Linux within next 1-2 years.

Regards
Mirek

Johnnyzee    10/07/2007
Mirek:
Thanks for your response. We will load our new software on XP OS for now and await the development of a Linux version.

## ID=873, Cumulative seepage face fluxes

Castanheira    10/10/2007
Dear Hydrus users

I'm making a simulation of water and solute(Na+) dynamics between one field with pipe drains, and the same field with pipe drains and mole drains (above pipe drains). I use a seepage face for each drain.

The upper limit is a variable boundary condition, and I have rain, irrigation and evaporation. When I run the model for 308 days, I obtain results which I can't understand. In both cases the cumulative variable boundary fluxes are the same. However I use 3 observation nodes and one of them at 30 cm of upper limit and in de middle of the drains, in this node I obtain higher concentration of Na+ with ( pipe and mole drain) than with only a pipe drains, Why? It should be the inverse!, I have a big seepage face in the first case, but the solution reveals cumulative seepage face fluxes higher in the case of only a pipe drain. What I'm doing wrong?

Thanks in advance
Paulo Castanheira


**ID=874, Two phase flow**

Tulip      10/10/2007
Hi,
I am a new user of HYDRUS-2D. Does HYDRUS solve single phase flow simulation or does it solve two-phase flow and considers the effect of pore air on infiltration?


Jirka    10/11/2007
HYDRUS considers single phase flow.
J.


Tulip    10/11/2007
Jirka,
Thanks for the reply. So that means when liquid is injected in dry soil in a small domain which is closed from the sides, the pressure heads near the injection pipe are experimentally found to be high initially , and then with the increase in degree of saturation of the underlying soil the heads would decrease. The progressive movement of the saturated water front in the soil is also found to be slow due to the resistance offered by air. HYDRUS will not be able to capture the delay or the initial high heads. The heads would match only when the soil is saturated and the increase in water content of the underlying soil would be within a very short duration in HYDRUS...right?

Thanks again
Tulip


Jirka    10/11/2007
HYDRUS basically assumes that the pressure in the air phase is always at a constant, atmospheric pressure. HYDRUS will not calculate build up of air pressure in front of infiltrating fronts, and similar effects.
Jirka

**ID=875, Source code**

Gisela   10/16/2007
Dear all,
we would like to change the seepage boundary condition to a value other than zero. We already managed to do this for the forward modelling. Is there a source code for the inverse version (h1d_clci.exe) available, too?
Thanks a lot
Gisela


Jirka    10/16/2007
No, we do not share it with the public. We share only the source code for the direct calculations, i.e., h1d_calc. Jirka


Gisela    10/16/2007
Hello Jirka,
we do not need the newest version. We would be totally satisfied by an old one. But I suppose, we can´t get this, either...

So, let me ask you one question: Do you see an other possibility to simulate a lower boundary condition that does not allow upward flow and has a constant head smaller than zero? We already tried to apply a simulated thin layer of something like gravel at the bottom, that more or less prevents the upward flow. But the results are not as good as with the "seepage-face version".

Thanks a lot,
Gisela


Jirka    10/17/2007
Gisela,
Send me an email and I will email you h1d_clci.exe (inverse code) that has a seepage face BC with a user specified pressure head. You need to enter this value immediately after you start the code when requested.
Jirka


 Gisela   10/18/2007
Dear Jirka,
thank you very much. I send you an email. It will highly improve and facilitate my simulations. It is great that - also for a public domain programm - you provide so much service.
Best regards
Gisela

**ID=876, Inverse solution - temperature data?**

mas422   10/23/2007
Hello,
I have thermocouple data for five depths in a canal which I am trying to model. I don't see any "type" number for putting this data into the inverse solution page. Is it possible to use inverse solution with temperature data, and if so, how?
Thanks,
Margaret


Jirka    10/23/2007
HYDRUS-1D can use temperature data for inverse simulations. You need to do the following:
a) When you select the inverse option in the "Main Processes" window, then in the "Inverse Solution" dialog you need to check "Heat Transport Parameters".
b) In "Heat Transport Parameter" dialog you need to select which parameters you want to optimize.
c) In "Data for Inverse solution" you need to give time (x), temperature (y), type (4), position (observation node number) and weight.

That's it.
J.


**ID=877, Column reactor experiment simulation**

Arao    10/24/2007
I'm new to Hydrus, and am trying to simulate some Ba breakthrough curves in upward flow, saturated sand column reactor experiments to obtain information about Ba sorption. I chose units of cm, hr, and mol.

I've looked at the examples, and am going through the questions, and have a few extra questions. I'd appreciate your help!

1. Initial, minimum and maximum time steps: I'd like to use a time step of 0.5 hr. Why do I need to specify min and max?

2. Water flow parameters (Qr, Qs, alpha, n, Ks, and I). I think here I need to enter porosity and flow rate. After reading the help section, none of these variables are porosity or flow rate. Where do I enter these? For flow rate, what units should I use? For saturated flow, what should I use for the other variables here?

3. Water flow boundary conditions. Initially I thought I should use Constant Flux at top and bottom, but in the discussion forum somewhere I read that I should use Constant Pressure Head?? And an initial condition in the pressure head??

4. Solute transport boundary conditions. I chose concentration flux at the upper BC and zero gradient at the lower BC (correct?). Looks like I have to specify a concentration at the upper BC...is this in units of mol/cm3(bulk) or mol/cm3(Water)?

5. Data for inverse solution. What "position" should I use for the data points? I measured Ba concentrations in the outflow of a 5 cm column.

Thank you!


Arao    10/25/2007
...another question about this same system:
As I mentioned above, I'm using units of mol, cm, and hr. The program solves for KD (sorption coefficient), among other things. What units are this in? I've read the manual and online help, and I think it's cm3/g. But are these cm3 of bulk sediment, or cm3 of water?
Thanks again.


Jirka    10/26/2007
1. If you want to simulate water flow, then HYDRUS will optimize its own time step. It will vary between the minimum and maximum allowed time steps. You can make all these values the same (0.5h), but there is no warranty that the solution will converge. If you run only solute transport (and it is linear), then you can fix the time step this way.

2. Qs- the saturated water content is the porosity. Ks can be the flow rate if you adjust the pressure gradient so that it is equal to 1. Darcy's law: Flow rate=Pressure head gradient * hydraulic conductivity

3. If you have constant upward flux, then fix the upper and lower boundary's pressures so that you can comply with the Darcy's law (given in 2).

4. IF the flow is saturated then it does not matter mathematically whether flow is upward or downward. Simulate it as downward flow and use concentration flux at the top and free drainage (or zero gradient) at the bottom.

5. Specify observation node at the outflow boundary, and if it is the only observation node, then position is 1.

6. Look at the governing equation and it will be clear.

Jirka


**ID=880, Numerical method**

Laurin    10/31/2007
Dear forum members,
Dear Jirka,
I tried to reproduce HYDRUS-1D results with a MATLAB script using the numerical method described by the manual and Celia 1990. However, HYDRUS-1D and the MATLAB script produce different results with van Genuchten hydraulic properties. I

fixed the time step within HYDRUS, so that adaptive time stepping is suppressed. Is there any difference between the Celia scheme and HYDRUS that would explain different results? E.g. I am not sure about the use of the internal interpolation table for K(h), C(h) and theta(h) that is used in HYDRUS.
Thanks for your help!
Laurin

Jirka     10/31/2007
Laurin,
The numerical solution in HYDRUS is not exactly as described by Celia et al., although the basic idea is the same. The exact solution used in HYDRUS is described in the HYDRUS technical manual.

You can get slightly different results from the exact solution when you use the interpolation tables of soil hydraulic properties since then we indeed just interpolate in these tables instead of evaluating exactly the function involved. You can disable the tables by specifying both limits equal to zero. Then you should get an exact solution if your discretization is sufficient.

We have recently discussed the issue of the interpolation tables in the following manuscript.
Zlotnik, V. A., T. Wang, J. Nieber, and J. Šimùnek, Verification of numerical solutions of the Richards equation using a traveling wave solution, Advances in Water Resources, 30, 1973-1980, 2007.

Jirka

Laurin   10/31/2007
Jirka,
Thanks for your quick reply. In my MATLAB program I follow closely the instructions from the HYDRUS technical manual. However, I get different results when the n-parameter of the van Genuchten model gets close to 1 (unfortunately the first case I tested). Otherwise results are indistinguishable. Could this be due to roundoff errors in HYDRUS? MATLAB uses double precision as far as I know.
Laurin

Jirka     10/31/2007
Have you disabled the interpolation tables?
J.

Laurin   10/31/2007
Yes, I set both limits for the tension interval to 0.

Jirka     10/31/2007
HYDRUS uses single precision, except for time and final matrices. But I would be surprised that this would cause some major errors. J.


**ID=881, Inverse solution and adsorbed phase example**

Mnimmer    11/01/2007
Greetings -
I have two separate, un-related questions I was hoping to get some thoughts on:
1) In trying to run an inverse solution for water flow (set to run 10 iterations), the solution stops as if it were set for zero iterations (as if it were solving the direct solution). In checking the output information, the solution did not stop because no further reduction in the objective function could be found - it simply stopped and gave a solution using my starting hydraulic parameters. The only change between this and other successful inverse solutions, is I increased the number of nodes/elements, and I added a slope to the initial water table.

2) I am looking for an example problem that simulates the desorbing of a contaminant from solids. I am trying to simulate a column (I could run this in 1-D, but I chose 2-D for the moment) with initial absorbed and dissolved - phase contaminants. There is no contaminant flux being added to the column, only a water flux through the top boundary condition. The only source for sustained contaminant flux leaving the column should be that desorbing from the solids. My simulations show that once the initial concentrations in the pore water flush through the system (this happens fairly rapidly), there is no sustained release from the solids. I decreased Kd and ran for very long time periods, but still no breakthrough at the end of the column, or even at observation wells placed within the system. If there is an example of this type of situation I may have missed, or if there are any suggestions of things to look at, it would be appreciated.

Thank you for the time.
Regards,
Mike


Jirka     11/01/2007
Mike,
1. Have you selected at least one parameter to be optimised? If you do not, then the solution stops after the initial run? I sometime do this error.

2. If you use only instantaneous sorption (i.e., the distribution coefficient Kd), then you specify as initial condition concentrations in the liquid phase (the code automatically calculates how much is adsorbed, i.e., s=Kd*c). The breakthrough outflow will simple reflect retardation (R=1+ro*Kd/theta). There should be no further concentration outflow after that. THis is what equilibrium model should predict.

If you want to have a longer concentration outflow, then you need a kinetic sorption/desorption model. Then you can specify also initial concentration for the

kinetic sorbed phase (variable Sorb). Then eveything will depend on the mass transfer coefficient (alpha). It can go fast, but also very slow (infinity for small alpha).

Jirka


### ID=882, First-order rate constant for solid phase

Tineke 11/12/2007 : 11:45:21
I am simulating the transport of pesticides through a column which contains a mixture of peat, sandy loam soil and straw. I simulated a column with and without degradation. However if I put my degradation constant in sinksolid1, I have a faster breakthrough than without degradation. I expected a same or latter breakthrough, but lower steady state concentration. How can this be explained?


Jirka 11/12/2007
Tineke,
I have run a quick test (steady-state flow) and saw exactly what I would expect to see, i.e., the arrival at the same time except at different (lower) concentration levels.
Jirka


### ID=883, Variable boundary conditions

Francesco 11/14/2007
Hi,i'm Francesco and use hydrus-1d for my job of university thesis "water flow in unsaturated soils".
I would want to know like assigning the just value for the parameter hCritA, and to know which variables influence it.
Please help me!
Thanks for your collaboration!


Jirka 11/14/2007
See decription on page 31 of the manual.
hA is minimum pressure head at the soil surface allowed during evaporation. When this pressure head is reached, the potential evaporation is reduced to get actual evaporation. The value for hA is determined from the equilibrium conditions between soil water and atmospheric water vapor.
J.


### ID=884, Dual porosity, input parameters

Gisela 11/16/2007
Dear everybody,
I am trying to use the dual-porosity option (mobile-immobile water with mass transfer) of Hydrus 1d and have a question about the parameters Qr and Qs: Do they

specify the total residual and saturation water contents (Qimmobile + Qmobile) or just the water contents of the macropore region?

I am asking, because my simulated Qs-values are always bigger than my QsIm-values, though I suppose that the pore-volume of the matrix region of my soil should be larger than the macropore-volume.
Thank´s a lot
Gisela


Jirka    11/16/2007
Gisela,
Qr and Qs are the residual and saturated water contents of the mobile domain, respectively.
QrIm and QsIm are the residual and saturated water contents of the immobile domain, respectively.

Total soil porosity should be Qs+QsIm.

If Mobile region accounts only for fractures or macropores, then it should indeed be likely smaller than immobile domain. However, in many applications mobile region represent not only macropores, but also mobile part of the matrix, and immobile zone only immobile domain, such as inside aggregate. Then the relationships can be reversed. This is only a mathematical model, and it can be subject to many different ways of interpretation.

Jirka

See the following article for exact definitions.
Šimùnek, J., N. J. Jarvis, M. Th. van Genuchten, and A. Gärdenäs, Review and comparison of models for describing non-equilibrium and preferential flow and transport in the vadose zone, Journal of Hydrology, 272, 14-35, 2003.


Gisela, 11/28/2007
Dear Jirka,
thanks a lot for your answer.
I would like to ask you one more question about the water contents: When I estimate the saturated water contents of the mobile and immobile region simultaneously, I sometimes get total water contents (mobile + immobile water content) larger than 1. How is this possible for volumetric water contents?

Best regards
Gisela


Jirka,  11/28/2007
Gisela,
The optimization program does not control whether the optimized values make sense. You need to use parameter constraints (max and min values) to prevent that.

I remember that I implemented in December 06 (and that should be in our version 3.03) the following option:
if(iModel.ge.6) ParD(8,M)=Por(M)-ParD(2,M)
Which means that for dual-porosity model (iModel.ge.6) the sum of the saturated water contents of the mobile (ParD(2)) and immobile (ParD(8)) zones is constant and equal to porosity (Por). The porosity is equal to the the sum of the saturated water contents of the mobile and immobile zones given by initial estimates. In this case you optimize only ParD(2) and ParD(8) is automatically adjusted.

Jirka


Gisela, 11/29/2007
Dear Jirka,
thank you very much for your prompt answer. I did not know that the clci.exe you sent me was modified in comparison to "normal" one and I always used the "normal" version for dual-porosity modelling. With the other version it works perfectly.
Best regards
Gisela


## ID=885, Input of concentration with no response

George    11/16/2007
Dear all,
I am simulating the transport of a chemical in soil. The application of the substance is made via the "variable boundary conditions table": A concentration of the substance is specified. Simultaneously, a precipitation amount of 0.01 (length unit is cm) is specified at this time point (one time point is a single day). The simulation is run over many years, with an application every 365 days.
I specified two observation nodes directly at the surface and at the following node.
In the output file (obs_node.out), the resulting concentration could be seen. Now, I have the phenomenon, that in the first six/seven years no concentration is responded in the observation nodes, although substance and precipitation are specified. In the later years, concentrations > 0 are given in the output file.
Increasing the concentration of the substance does not change this.
Increasing the precipitation amount (to 0.05) on the other hand has the effect of concentrations > 0 after all applications.

Where is the substance "left" when the precipitation is only 0.01 cm? Has somebody an explanation for me?
Thank you!


Jirka    11/16/2007
Joerg ,
The amount of solute applied is S=cTop*Prec. However, Prec must be larger than Evap as the surface flux is qTop=Prec-Evap. If Evap is larger than Prec, then there is

not infiltration and no solute enters the soil. When there is Evap at the same day, but Prec is larger than Evap, then

ConcNew=Prec/(Prec-Evap)*cTop
S=ConcNew*(Prec-Evap)

Jirka


**ID=886, Cumulative fluxes at an observation node**

George    11/16/2007
Dear all,
I need the cumulative fluxes of water and solute at a specified observation node. The manual says - for my understanding - that no cumulative values are given for user specified observation nodes, only for the top and bottom of the profile (when T-Level-information is checked on).
Is that right?
Can I derive the cumulative values for my observation nodes from other values?
Alternatively, I could use the cumulative concentrations for the observation nodes, but they are given not at all!!??

Thank you for help!


Jirka    11/16/2007
If you send me an email, I could send you a version of the code in which we print into the obs_node.out file instead of temperatures, water fluxes (we will certainly make it an option in the next version as many people request this). You could then easily integrate these fluxes over time to get cumulative water fluxes.
What is cumulative concentrations?
J.


**ID=887, Precipitation entry**

vlhauser 11/20/2007
I am modeling lysimeter data with HYDRUS-1D. I entered precipitation data for a 2-year period. The output to the file "T_LEVEL.out" contains the variable "sum(Infil)". The description of the output files says that sum(Infil) is cumulative infiltration. Since, rainfall is the only input, I presume that sum(Infil) should equal precipitation entered in the time variable boundary file. The total sum(Infil) at 731 days is one-half the entered precipitation. Where can I find precipitation in the output file?


Jirka, 11/21/2007
Precipitations are given in the input file (Atmosph.in). In the output file, I give the potential surface flux (rTop, which is precipitation minus evaporation) and then the actual flux (vTop), as well as their cumulative values. If cumulative infiltration is

smaller than cumulative precipitation, then you likely had precipitations larger than Ks at some times, which resulted in surface runoff (another column in this file).
Jirka


Vlhauser,  11/26/2007
Thank you for your prompt response.


Vlhauser,  11/30/2007
I was unable to determine how to change model input to get good output from the previous reply. I will restate the problem hoping to improve clarity in communication.

Alfalfa modeling, Lysimeter, Bushland, Texas
I am using HYDRUS1D. Alfalfa was established before the beginning of the measurement period. I am modeling 2 years (Jan 1, year 1 to Dec. 31, year 2). The potential evapotranspiration is very high at the site. Potential ET calculated by Penman-Monteith using weather data from the site resulted in PET up to 2.0 cm/day during year 1, and 44 days of PET > 1.0 cm/day. Typical wintertime PET ranges from 0.1 to 0.5. The alfalfa was irrigated to meet water demands. The precipitation at the site and irrigation were both included in the daily "precipitation" input to the model. Potential evapotranspiration (PET) was computed by the Penman-Monteith method, for each day of the 731-day modeling period. Daily values of evaporation and transpiration entered into the model were computed from daily PET using the surface-cover fraction method and daily values of LAI computed by the EPIC model for each day of the 731-day modeling period.

The model results compare to measured amounts as follows (units are cm):
Precipitation measured = 295.3, sum(infil) = 147.6;
ET measured = 302.8, sumvroot + sum evap =177.6;
Surface runoff measured = 0, runoff = 0;
Deep percolation measured = 0, sumvbot = 0.008.

I have the following questions:
1. Is "Sum(infil) the sum of input precipitation?
2. Is the sum of vroot and evap supposed to equal measured ET?
3. Why is the model output for precipitation only half the input value?
4. How should I enter the data to obtain valid results?


 Jirka,    11/30/2007
1. Is "Sum(infil) the sum of input precipitation?
Yes, unless the program generates the surface runoff. Input value is the potential infiltration and the code calculates the actual infiltration. The difference between those two is the surface runoff.


2. Is the sum of vroot and evap supposed to equal measured ET?
Not necessarily. You enter potential values and the code calculates actual values (root uptake reduction due to either too wet or dry conditions, evaporation reduction due to too dry conditions).

3. Why is the model output for precipitation only half the input value?
It should not if everything goes all right. You should check the mass balance errors. If they are larger than 1%, then there was a problem. If they are less than 1%, then there must be some explanation.

4. How should I enter the data to obtain valid results?
Check the text examples, e.g., test 2 for a proper set up.
Recommended values:
Spatial discretization: about 1 cm or less at the top.
Water content tolerance: 0.001
PRessure head tolerance: 1 cm
Initial time step: 0.0001 d
Minimum allowed time step: 1e-7 d

Jirka

Vlhauser,   11/30/2007
Jirka, thank you for your prompt reply. Now I know that I have a problem. I will check the mass balance errors. I will also recheck the test2 example to see if I missed something and verify that I have the numbers from your answer no. 4 correctly entered.
In question no. 2, I am comparing vroot plus evap (actual values) with measured actual values. Therefore, I assume that I made a proper comparison. The significant problem with precipiatation may be the source of the vroot plus evap error.
Thank you for your help.

Vlhauser,   12/06/2007
The mass balance errors were all less than 0.5%. None of the recommended values improved the situation. I discovered that the problem was caused by not using "Heat transport" in the "Main Process" file. When I checked that box, the program worked as expected.

I did not use "Heat transport" because I could find nothing in the instruction manual or on-line stating that this is necessary or any discussion tying "heat transport" to estimation of water flow. When I entered my first data set, I checked "Heat transport"; however, when I came to the "Time Variable Boundary Conditions", there was no column in which to enter air temperature data. Therefore, I assumed that it was not needed and removed the check from the "heat transport" box.

The instructions or help should clearly state
1. what is needed as input to calculate water flow and water balance, and
2. How to get space for air temperature in the "time variable boundary conditions" input page.

Jirka   12/11/2007

Your conclusions are not correct.

a) You do not need to run heat transport simultaneously with water flow calculations unless you check that you want to consider temperature dependence of soil hydraulic properties (the same is true for solute transport).

b) If you do not check "Temperature Dependence" of soil hydraulic properties, then water flow is completely independent of heat transport and temperatures in the soil. Thus heat transport can not affect your water flow simulations.

c) tTop (surface temperature), tBot (bottom temperature) and Amp (daily amplitude of surface temperatures) columns are automatically generated in "Time-variable boundary conditions" window when "Heat Transport" is checked and "Time variable boundary conditions" are selected.

Jirka

Vlhauser, 12/20/2007

The reply on 12/11/2007, states that "Your conclusions are not correct." The following are more results of model output.

All of my models have been run with "Temperature dependence of soil hydraulic properties" turned OFF.

A model run called "Heat Transport" had heat transport turned ON. The "Heat Transport" file was saved under a new name "NO Heat Transport", then the heat transport box within the main process window was unchecked in the new file. The resulting answers were as follows:

"Heat Transport" produced precipitation with error of 1.7%, and evapotranspiration with error of 3.9%. "No Heat Transport" produced precipitation with an error of 50 % and ET with an error of 43%.

If "Heat Transport" does not need to be checked, please explain what is going on.

Jirka   12/20/2007

I would need to see the files of the project to be able to find out what went wrong.
Jirka

**ID=888, Perched initial pressure heads**

Brian, 11/21/2007

Good Day,

I am creating a 3D model with two perched initial pressure head conditions (H = 0 m) underlain by a third initial pressure head condition (H = 0 m), which exist due to vertical heterogeneity in material hydraulic properties (gravel overlying glacial till overlying silt overlying shale). Moving from bottom to top in the model domain, initial pressure heads will be 1 m (at the bottom) decreasing to, say, -5 m (at the top of the silt), then 0 m decreasing to -5 m (at the top of the till), and again 0 m decreasing to -5 m (at the top of the gravel). Also, the value of - 5 m was picked arbitrarily.

The initial conditions will extend completely across the model. The top of the model will be an atmospheric boundary or variable flux boundary (I am modeling the effect of open pit drainage on the water table(s)). The bottom will be a no flow boundary. Model dimensions are approximately x = 300 m, y = 300 m , z = 50 m.

I assume that this is a relatively easy suite of initial conditions to create, but if anyone knows of any issues that may arise in this scenario, I would appreciate learning about them.

Cheers,
Brian


 Jirka   11/22/2007
Brian,
Perched water layers are one of the most difficult things to solve with variably-saturated codes. This is because this phenomenon is usually caused by sequence of materials of dramatically different soil hydraulic properties (clay layers between sand layers) and because there are usually dramatic changes in pressure heads and water contents at interfaces between different materials. The code can accommodate that, but it requires rather small discretization at the interface between material layers. I have done simulations for 2D cross-sections several hundred meters deep and wide, but I had almost 50k elements in that simulations. Thus in 3D the number of required elements may be excessive for a PC to handle. I can see that your domain is relatively large. Can't you simplify your problem into a 2D to make it more manageable?

Jirka


Brian   11/26/2007
Hello Jirka,
Thanks for the response. I agree that a 2D representation of the site would be a much more reasonable numerical approach. Depending on the requirements of the client, I may be able to pursue this approach. Some of our clients are beholden to 3D models, which as you have iterated are not always workable in variably saturated, heterogeneous, multiple perched water table conditions.

Thanks much,
Brian


**ID=889, Initial water content is lower than Qr**

yxb967,   11/25/2007
Hello,everybody
I'm modeling winter wheat ET from a shallow phreatic water table(130cm below the surface)in inverse solution.HYDRUS ceased with DOS indicating "initial water content condition is lower than Qr" .But increasing the Minimum water content in the surface of initial water content profile is not helpful.

The screen show "initial water content condition is lower than Qr" what it mean ?
How i can solve the question?
I'll appreciate you for your help!


eagle-zhao, 11/25/2007
Hi yxb967,
Maybe you should notice the value of Qr (residual water content) in hydraulic parameters and make it lower than the initial water content. In HYDRUS, initial water content (any other water contents) can not be lower than the residual water content.

Eagle-Zhao


yxb967,  11/25/2007
Thank you very much for giving me a so quick answer!
Indeed the initial water content of all the profile(Moisture content ranged from 0.1 to 0.43 (v/v) of linear distribution ) is higher than Qr! I don't understand why the screen showed "initial water content condition is lower than Qr"
When I make Qr lower to 0.015,the problem is solved! Inverse solution information(in post-processing)gaved Qr=0.0015923 with Non-linear least-squares analysis in 95% Confidence limits.Is that right and practical to loam?(Hydrus gives the minimum of Qr is 0.034,the difference is huge)
Thanks!!


yxb967,  11/25/2007
How to assign to Feddes' Parameters in root water uptake paramerers properly?
another question is whether r2H and r2L must be set at 0.5cm/d,0.1cm/d respectively?the ET is higher than 0.5cm/d Prevalently!
Are there related literature ? I don't get help from the manual and Application Help.

thank you very much!
yxb967


Jirka    11/25/2007
If you run inverse simulation and optimize the residual water content, you need to constrain it so that the inverse routine can not choose value larger than your initial water contents (if initial condition is specified in water contents).
Jirka


yxb967   11/26/2007
Hi Jirka
Now, I understand the crux of the problem
thank you very much for your very professional reply!
yxb967

## ID=890, Feddes' parameters about winter wheat

yxb967,  11/26/2007

Hi everybody

How to assign to Feddes' Parameters in root water uptake paramerers dialog box properly in winter wheat ET simulation ?

Another question is whether r2H and r2L must be set at 0.5cm/d,0.1cm/d respectively?the ET is higher than 0.5cm/d Prevalently!

Are there related literature ? I don't get help from the manual and Application Help.

I will be very grateful to you!

yxb967


Jirka, 11/26/2007

Make your own literature search. We have entered in the HYDRUS-1D database whatever we could find.

J.


## ID=892, How to increase ETp time of variable BC

yxb967  11/28/2007

Hi everybody

I'm modeling winter wheat ET data from shallow groundwater. The ETp (according to FAO)is lower than the ET of Experimental observations because no wheat growing surrounding the test site which is 1m¡Á1m square leads the marginal effect! How should I deal with time variable BC Evap.and Transp.?


Jirka 11/28/2007

I guess, just increase both values due to this effect. But how much, I do not know. I hope someone else can help with this. J.


yxb967,  11/29/2007

Hi,

Anybody who knows the answer tell me what the answer is£¬please£¡urgently needed!

Thanks!

yxb967


eagle-zhao,  11/29/2007

I don't get well with your problem. Normally, you need in situ measurements to partition evaporation from evaportranspiration. In addition, the ratio between potential transpiration and soil evaporation can be estimated based on radiation partitioning according to Beer¡¯s law as a function of LAI.

Zhao

yxb967, 11/29/2007
hi,eagle-zhao
I'm sorry that I didn't make myself understand because of my lower English mastery level!

I don't want to know how to partition evaporation from evaportranspiration! My observation evapotranspiration value is larger than the value of potential evapotranspiration ETp( the Etp into two for Evap.and Transp. being assined to Hydrus as Atm. BC).
I want to know how to make ETp larger to accord with my field data!
I hope I can ecpress myself clearly!

Any good advice?
Thanks a lot! Anxious wait for a reply


eagle-zhao, 11/29/2007
For this, I haven't definiting answer. Hope following reviews is helpful.
How do you get the reference ET from FAO functions? Did you measure the parameters of FAO in situ or what else?

What time did you measure the ET (e.g. the soil mosisture condition and the phase of crop growth at that momment) because it is highly related with your measured results.

If you consider the marginal effect, maybe you can find some checking information from the handbook of FAO if I remember right.

In HYDRUS, you must partition Etp into two for Evap.and Transp. How do you do it?

If you assure the both FAO reference and measured ET data, maybe you can transfer each other directly. HYDRUS will calculate it not only from your input data but also from actual soil water, root density etc..

Zhao


yxb967, 11/29/2007
I have computed the reference ET (ETo),ETp(=K*ETo,single crop coefficient method),ETc( = Kcb * ET0,dual crop coefficient method),ETs£¨= Ke * ET0,dual crop coefficient method£©from FAO functions according to meteorological datum£¨Temperature,vapor pressure,wind speed etc.),LAI£¬soil moisture content and the phase of crop growth etc..This is no doubt!

According to FAO functions "ETc = Kcb * ET0 "and "ETs = Ke * ET0" ,I import Evap.and Trans.as variable atmospheric BC not using formulas "transpiration = evapotranspiration * Leaf Area Index" and "evaporation = evapotranspiration * (1-Leaf Area Index) ".is it correct?

Also I want to get a method to increase Evap.and Trans. scientificly considering the marginal effect£¨The soil mositure condition ,Evaporation from shallow groundwater table and the phase of crop growth available£©.

thanx!
yxb967


eagle-zhao,  11/29/2007
For the marginal effect, that seems something specific with detailed case.

Some reference information followed:

Hi Jirka
could you explain how to split evapotranspiration in transpiration and evaporation?
Is it a simple process like this reported below?

transpiration = evapotranspiration * Leaf Area Index
evaporation = evapotranspiration * (1-Leaf Area Index)

thanks for your help


No, it is not that simple. You need to first calculate the "Surface cover Fraction" from LAI:

SCF=1.-exp(-rExtinct*LAI)
rExtinct=0.463

and then

transpiration = evapotranspiration * SCF
evaporation = evapotranspiration * (1-SCF)

Jirka


**ID=894, Positive or negative?**

yxb967,  11/29/2007
In Geometry Information and Root Growth Parameter(Pre-processing) Depth of the Soil Profile and Initial Rooting Depth(Maximum Rooting Depth)should be assigned to positive or negative,respectively?
In Soil Profile the soil surface is defined zero, below it negateive,but in manual the bottom of the soil profile is located at x = 0 and the soil surface at x = L (P46)!

I'm puzzeld! Anyone could interpret this?
Thanx~
yxb967

**ID=895, Output of Brook-Corey**

Srilert, 12/03/2007
Dear Sir,
I wondered when I used RETC to estimate retention parameters using Brook-Corey model. I found outputs are ThetaR ThetaS, Alpha, n which are Van Genuchen model not Brook-Corey.
How I get the parameters of Brook-Corey model.
Could anyone suggest this ?
Looking forward for your answers.
Best regards,
Srilert


Rvang,   12/04/2007
Srilert:
alpha equals 1/(air entry value), and n is the pore-size distribution index in the BC equation.

Regards. --Rien van G.


**IF=896, CXTFIT output**

Alessandro, 12/03/2007
Dear all,
I am very new to STANMOD/CXTFIT, and trying to understand how to use it. I am a bit confused by the output format when using the deterministic non-equilibrium CDE. Particularly, I do not understand the meaning of the variables C1 and C2. According to the manual, these are the concentrations for the equilibrium and non-equilibrium sites (page 17 of the manual). How are these concentrations related to the fluid concentration at the sampling location?
Also, I would expect the value of C1 to be 0 when I set f=0 (non-equilibrium sites only). This is not the case (see e.g. case 1, example 7.6a). Could you please give me some hints on this?

Thanks!
Regards,
Alessandro


Jirka,   12/03/2007
Alessandro,
Here is a quick answer (I'm sure Nobou will give you a more detailed answer later):
C1 is the concentration in the equilibrium phase. This means this is the concentration in liquid (mobile) phase (not on equilibrium sorption sites).
C2 is the concentration in the nonequilibrium phase. This means this is the concentration either in the immobile liquid phase or on the kinetic sorption sites.
For the dimensional forms of both C1 and C2 concentration consult the manual.
Jirka

Ntoride,   12/06/2007
Alessandro:
Thanks, Jirka, for your prompt reply.

>Also, I would expect the value of C1 to be 0 when I set f=0
>(non-equilibrium sites only).

As defined in Table 3.1 of the manual, $C2 = s\_k/(Kd\ c\_o)$ for the one-site model assuming f=0. C2 is the adsorbed concentration of the kinetic site and C1 is the concentration of the solution phase for the chemical nonequilibrium model. Please note that we use the same nondimensionless equations for physical and chemical nonequilibrium transport. For detailed information, please refer to

van Genuchten, M. Th., and R. J. Wagenet, Two-site/two-region models for pesticide transport and degradation: theoretical development and analytical solutions, Soil Sci. Soc. Am. J., 53, 1303-1310, 1989

Please also note that the difference between the flux concentration for effluent and the resident the concentration:

Parker, J.C., and M. Th. van Genuchten, Flux-averaged and volume-averaged concentrations in continuum approaches to solute transport, Water Resour. Res., 20, 866-872, 1984a.

If any further questions, please do not hesitate to ask me.
Nobuo


Alessandro,  12/11/2007
Dear Jirka, dear Nobuo,
thanks for the very prompt reply. There is still something I don't understand, but I am reading the references suggested.
In case I'll need more details, I'll post again on the forum.

Thanks a lot!
All the best
Alessandro


### ID=897, Brook-Corey in RETC

Srilert,   12/04/2007
Dear Sir,
I am sorry that post in this because I found that in RETC forum really few people to join, so, may be some can help me in this proplems.
I wondered when I used RETC to estimate retention parameters using Brook-Corey model. I found outputs are ThetaR ThetaS, Alpha, n which are Van Genuchen model not Brook-Corey.
How I get the parameters of Brook-Corey model.

Could anyone suggest this ?
Looking forward for your answers.

Best regards,
Srilert

Rvang, 12/04/2007
 Srilert:
When using the BC option in RETC, alpha in the output is 1/ha, where ha is the air entry value. n in the output the pore-size distribution index in the BC model. Please see the manual.
Regards. --Rien van G.

## ID=898, Value of hcrita

Francesco, 12/04/2007
i'm studying water flow in unsaturated soils.for example if i've:

Ks= 0.1
precip.= 252.56 mm/h; 25.256 mm/h; 2.5256 mm/h
Evap.=0
hCritA=?

## ID=899, Export of multi-layered geometry

Luisa, 12/05/2007
Hi,
I built a multi-layered 3D domain by importing x,y,z-coordinates. I then rotated the domain to obtain a steeper slope. I would now like to extract the x,y,z-coordinates from this new rotated domain. But when I use "export geometry to a text file", only the coordinates for the points at the surface and the lower boundary of the domain are exported not any inter-layer points. Is there a way to also get those coordinates?

Thank you, Luisa

Mirek, 12/06/2007
Hi Luisa,
Unfortunately, there is no direct way to export these coordinates. These inter-layer points are calculated from points at the base surface and z-coordinates defined at individual thickness vectors. There are two ways to get needed coordinates but both are rather difficult:

1/ You can create a simple FE-Mesh (FE-Mesh -> advanced options -> Fundamental triangulation + 3D mesh with 1 domain layer = 1 FE-Mesh layer) and then save this mesh as a text file. Then you will find required nodal coordinates in this file.

2/ I remember that you imported your domain from a text file with many thickness vectors, each one with multiple z-coordinates defining inter-layers. I think that you could calculate required coordinates for example in Excel if you export rotated points laying in the base surface and then add z-coordinates of thickness vectors defining your inter-layers.

I have a question: Why do you need these points? Would it be useful for other HYDRUS users to be able to export these points?

Regards Mirek


Luisa, 12/06/2007
Hello Mirek,
the reason why I am interested in exporting these points is that I would like to calculate the topographic index which needs calculation of flow direction, flow accumulation and slope. It can be done in ArcGIS but I would need the DEM for that. I investigated the generation of subsurface flow depending on slope and there are some interesting interactions between slope and topography of the bedrock surface (which are in my case the inter-layer points). It would be great to support my observations with maps of the topographic index.
I might try one of the methods you suggested.

Thank you, Luisa


**ID=900, Help! How thinking ponding at the upper boundary**

Zhoujian, 12/06/2007
I am a beginner at Chinese Academy of Science's .
I have a question? I do the rain infiltrate experiment of soil column, but the upper boundry condition thinking ponding I did not know how should choose?
For example, I continue 35 minutes by the 0.4cm/min rainfall amount simulation. Starts the ponding at the 4th minute. Ponding process as follows: 0.5cm at the 5th minute, 1cm at the 8th minute, 1.5cm at the 10th minute, 2cm at the14th minute, 2.5cm at the 18th minute, 3cm at the 21st minute, 3.5cm at the 24th minute, 4cm at the 28th minute, 4.5cm at the 32nd minute, 5cm at the 35th minute (simulation rains conclusion); From this time water depth start to descend. The process as follows: 4.5cm at the 37th minute, 4cmat the 39th minute, 3.5cm at the 41st minute, 3cm at the 43rd minute, 2.5cm at the45th minute, 2cm at the 47th minute, 1.5cm at the 49th minute, 1cm at the 51st minute, 0.5cm at the 53rd minute, 0cm at the 55th minute. Then becomes 0 flux boundary condition. How should I establish such boundary in hydrus?
Very thinks! My e-mail: zhoujian@lzb.ac.cn


Jirka, 12/06/2007
Dear Zhoujian,
In HYDRUS (2D/3D) we assume that once applied flux (precipitation or irrigation) is higher than the soil infiltration capacity (zero pressure is reached at the boundary, i.e.,

ponding), all excess water is instantaneously removed by surface runoff. HYDRUS (2D/3D) can not account for water ponding (accumulation) at the soil surface, unless you specify water ponding at the soil surface directly using (positive) pressure head boundary conditions. This is because we do not consider in HYDRUS (2D/3D) overland flow. Thus if you want to have ponding at the soil surface, you need to specify the upper BC as "Variable Head BC" with values you have. You need to use the "Boundary Condition Options" "Interpolate variable pressure had in time" (to get continuous changes of water depth in time). Then you can compare calculated infiltration flux with your applied flux (or use the inverse HYDRUS option to calibrate the code).

However, you can do this with HYDRUS-1D. In HYDRUS-1D if you select "atmospheric boundary condition with surface layer", the code uses applied flux, calculates the infiltration flux and, if lower than applied flux, allows excess water to accumulate at the soil surface (and infiltrates is after precipitation stops). Thus in HYDRUS-1D you can specify flux (in your case 0.4 cm/min for 35 minutes and zero after that) and the code will calculate resulting ponding. You can then compare HYDRUS results with your measurement (or use the inverse HYDRUS option to calibrate the code).

Please note that I will be out of office for the next 10 days.
Jirka


**ID=901, Random walk particle tracking**

Botros,  12/07/2007
Hi Jirka,
I'm trying to run a transport model using a random walk particle tracking technique. I ran the flow problem in Hydrus until a steady state is obtained. I obtained the steady soil moisture velocity at each node (Vx and Vy) by dividing the darcian flux vector (V.OUT) by the water content values (TH.OUT). I used that steady soil moisture velocity in my particle tracking code. The results look unreasonable so I wonder if I'm missing anything in calculating the soil moisture velocity? I read in a previous thread that the file V.OUT might not be accurate especially in boundary nodes. Is that still an issue in Hydrus higher versions?

Thanks,
Farag


Jirka, 12/12/2007
Farag,
I'm calculating in all my codes velocity in the same way. As explained in the manual, I calculate pressure head gradients in all elements (each element has a single gradient because pressures are assumed to change linearly over the element), then take the nodal average gradient (average gradients over all elements containing a particular node) and multiply it by the nodal hydraulic conductivity (corresponding to the nodal pressure head). This is not the most precise method, but in general works reasonably well (I'm using these velocities in my solute transport modules). There were quite a

few papers in which my codes were coupled with particle tracking codes for evaluating solute transport in heterogeneous media (see the list below). It is true that the precision of fluxes are less precise than calculated pressure heads or water contents, because pressure heads are primary variables, with their values resulting directly from the solution of the Richards equation, while fluxes are secondary values, calculated from the primary values (pressure heads and conductivities). If the method fail for you, you may use for boundary nodes instead of fluxes from v.out, fluxes from Boundary.out, which are exact, i.e., results of the Richards equation solution.

Jirka

Hammel, K., and K. Roth, Approximation of asymptotic dispersivity of conservative solute in unsaturated heterogeneous media with steady state flow, Water Resour. Res., 34(4), 709-715, 1998.

Roth, K., Steady state flow in an unsaturated, two-dimensional, macroscopically homogeneous, Miller-similar medium, Water Resour. Res., 31(9), 2127-2140, 1995.

Roth, K., and K. Hammel, Transport of conservative chemical through an unsaturated two-dimensional Miller-similar medium with steady state flow, Water Resour. Res., 32()6, 1653-1663, 1996.

Tseng, P.-H., and W. A. Jury, Comparison of transfer function and deterministic modeling of area-averaged solute transport in a heterogeneous field, Water Resour. Res., 30(7), 2051-2063, 1994.

**ID=902, Subsurface lateral flow modelling in the field**

Shoebodh, 12/13/2007
Dear forum members,
I am a master student in soil and water science. I have been working in a project in which we have installed 90 wells( one meter deep and about 10cm diameter)in diverging in all direction from the northeast corner of the field( 10acre area). At the north east corner we injected KBr and collected soil samples for 6 weeks and analyzed the water samples for bromide concentration. I am also studying the saturated hydraulic conductivity of the profile upto 120cm depth. Now I though I could use Hydrus 2D to trace the movement of Bromide in that field and also model the subsurface lateral flow of water. The field supposedly has a hard pan at about 110cm depth which virtually restricts water movement downwards thus encouraging the lateral movement.
Please give you valuable guidances and ideas on this. If necessary I can give further details of the project also.

Hope I will get good guidance from you.
Thank you

Jirka, 12/16/2007
It is not clear what type of assistance you request. From your description it would seem that three-dimensional solution of HYDRUS (2D/3D) would be more appropriate if the bromide source is a point source.
Jirka


**ID=903, Non-equilibrium sorption**

George, 12/14/2007
Hi
is it possible to define different degradation rates for the equilibrium and the non-equilibrium phase when chemical nonequilibrium option is considered? Or is it possible to switch off the degradation in the non-equilibrium phase?
Thanks a lot!


Jirka, 12/16/2007
Joerg,
It is not possible to do that in the publicly available version of H1D. However, I have a version in which one can define degradation parameters to depend on water content, and this dependence can be different in the mobile and immobile zones (to account for example for nitrification and denitrification). By using this dependence you can manipulate the degradation coefficients any way you want. Send an email directly to me and I will send you the file.
Jirka


**ID=904, Bottom flux**

yxb967, 12/24/2007
hello,everybody
I'm modeling winter wheat evapotranspiration from shallow groundwater.But the simulation results show that Bottom Flux (evaporation from shallow groundwater) is too gentle ups and downs vs observed value.What's wrong with it?
I'll appreciate you


yxb967, 12/24/2007 : 04:17:05
Add a few words:
The boundary condition is Atm. BC: Evap.and Tran. is divided from ETp with significant fluctuations in value


Jirka, 12/24/2007
Whether soil can deliver enough water from the groundwater to satisfy the evapotranspirative demand will strongly depend on soil hydraulic properties and the depth of the groundwater.
Jirka

 yxb967,  12/27/2007
Dear Jirka, Thank you
But I don't get the satisfactory results whatever I adjust the modle.Can I give you the model?Please help me look at where the problem£¿


## ID=905, Happy new year to everybody

j0ku89,  12/29/2007
Lots of Success, Health, Happiness and $$$ to All!


## ID=906, Theta

Nacer,  01/03/2008
Heloo all,
I am simulating only water movement in a soil column with precipiation and evaporation,
my question is:witch value of theta can we put in the dialogue"soil profil summary" knownign that i have 03 differents material in the soil profil.
Thank you very much
Nacer


Jirka,  01/03/2008
You need to enter for each horizon its own initial condition (water content). Note that we always recommend to enter the initial condition in terms of the pressure heads, as the pressure head is the driving variable for flow in heterogeneous soils.
Jirka


Nacer,   01/03/2008
Think you very much for your response


## ID=908, Constraint of dispersion

Sophia, 01/04/2008
Dear Jirka
I am simulating Br transport in a lysimeter using Hydrus-1D. The depth of lysimeter is 0.7m. I would like to know what is the constraint (Max) of longitudinal dispersion.
Thanks.
Sophia


Jirka, 01/04/2008
Usual recommendation is to have dispersivity of one tenth of the length, i.e., 7 cm. I'm not aware of upper constraints, but obviously you can not determine dispersivities larger than 70 cm from data on a 70 cm column.
Jirka

Sophia, 01/05/2008
Thanks, Jirka. It is very helpful.


**ID=909, Sinkgas1'**

Sophia, 01/04/2008
Dear Jirka
I am trying dual-porosity Model (mobile-immobile, head mass transfer)in Hydrus 1-D for bacteria transport. It seemed going alright, but I found that whatever value I set for Sinkgas1'(initial estimate), the output result won't be effected by it, and that parameter just kept the initial estimate in optimized result. Is that normal? Thanks.
Sophia


Jirka, 01/04/2008
Sophia,
SinkGas1 is the first order degradation coefficient representing degradation in the gaseous phase. If you have no solute in the gaseous phase, i.e., your Henry's constant is equal to zero, then the solution is independent of SinkGas1.
Jirka


Sophia, 01/05/2008
Thanks, Jirka.
Sorry,what I mentioned in last question should be Bromide ( I wrote as bacteria by mistake). Sure, my "henry" set as zero.


**ID=910, Modeling in many years**

yxb967, 01/05/
Hi,everybody
I have two questions in modeling winter wheat ET from shallow water table.
1. The actual sufer flux as the modeling result is always negative when I don't consider precipitation and irrigation in the model boundary condition. Is that right?
2.If I want to model the 3th,4th,5th months of about twenty years in one entire model,how should I define the dates(that is to say how to input Time in BC )?I am puzzeld because I have considered root growth! Or model can be computered only in single year?

Thank you very much!
yxb


Jirka, 01/05/2008
1. Fluxes in HYDRUS-1D are positive upwards and negative downward. Thus precipitation and infiltration fluxes should be displayed as negative surface fluxes, while evaporation as positive surface fluxes.

2. HYDRUS-1D does not recognises Julian days, i.e., it does not start counting from zero every year. The time flows from zero on the first year, via 365 on the second, 730 for the third year, etc. The root growth function can be used only once and thus can not be used for multiple years. If you want to simulate multiple years, you need to have constant root depth and manipulate surface fluxes by dividing potential evapotranspiration into potential evaporation and transpiration. We are planning to upgrade HYDRUS-1D with the option to allow multiple season for the root growth, but do not have that ready yet.

Jirka


yxb967,  01/06/2008
To the question 1:
Yes, but I only consider evaporation and transpiration not precipitation and infiltration in my simulation.So I am suprised to the negative fluxes!
yxb


## ID=912, Mesh generation problem

Brian, 01/07/2008
Good Day,
I am having a problem with mesh generation in a 2D model. My model has three surfaces with irregular boundaries. The boundaries were manually entered as points from which polylines and then surfaces were generated. The problem which is occurring is that the mesh being generated does not take into account the irregular surfaces, but instead creates straight lines from one vertical side of the model to the other vertical side.

The green colored nodes which the model generates at the surface boundaries just before mesh generation create straight lines from one vertical side of the model to the other, with occasional, single green nodes placed at those points where I defined the polyline. I was expecting that the green nodes would instead create straight lines that went from each of my defined points on the polyline to the next defined point.

How can I remedy this problem?
Thanks much.
Brian


Mirek,  01/07/2008
Good day Brian,
I'm not sure if I understand correctly to your problem. Please send me the project, I'll look at it.
Regards
Mirek

**ID=913, Request for help on modelling uranium BTCs**

Elise,  01/09/2008
Hi,
My student conducted tracer experiments at our field site using uranine water tracing dye and bromide. The field site is for managed aquifer recharge using treated effluent and the tracer experiments were for estimating the travel time through the 10m-thick unsaturated (sand) zone. The recharge rate is about 17 litres/minute.

We have several nice datasets of uranine BTCs that were obtained from sampling from suction lysimeters at several different locations and 5 different depths in the unsaturated zone.

My reason for writing is to ask if anyone could please advise us on which code would be most suitable to analyse the data. We thought about using CXTFIT to obtain estimates of v and D, but I realise now that it may be unrealistic given that it is an advection-dominated system. I started by reading the STANMOD and CXTFIT manuals, focusing on Chapter 7 and the inverse sections. Several years ago I took a shortcourse on HYDRUS-2D, but really I don't have a lot of experience using it, so it would be nice to communicate with a more experienced person(s) on these topics.

We'd first like to know whether we should be aiming to use Hydrus-1D or CXTFIT.
It's not clear to me how one would input the water flux in CXTFIT if we took this approach. I could probably obtain rough estimates for the soil hydraulic properties and attempt using Hydrus-1D, but I was hoping to keep this simple as a first attempt.

If anyone has an example that sounds similar to our problem, could they please share it so that we can understand how to set up the problem?

Thank you,
Elise


Ntoride, 01/09/2008
Dear Elise: CXTFIT is based on analytical solutions for the CDE subject to steady state water flow with uniform water content. Hence CXTFIS is generally used for well-controlled experiments to determine solute transport parameters. HYDUS numerically evaluate the CDE with Richards equation. Although HYDRUS could be used for solute transport with transient water flow, it is necessary to give proper parameter vales for flow and transport with appropriate initial and boundary conditions. If any further questions, please do not hesitate to ask me.

Nobuo


Jirka,  01/09/2008
Elise,

I will just briefly expand on Nobuo's answer. Is not clear from your emails whether the water contents and water fluxes were constant during the tracer experiment. If yes,

and if you can assume that the soil profile is homogeneous, then you should be able to use CXTFIT (STANMOD). If answer to any of these questions is no, then you need to use HYDRUS-1D.

Jirka

Elise, 01/10/2008
Jirka and Nobuo,
thank you for your prompt replies.

It sounds like we could use CXTFIT (STANMOD), assuming that our sand is homogeneous and the water contents and fluxes are roughly constant. These are safe assumptions for several datasets obtained at shallow depths.

I'd be curious to compare with Hydrus-1D.

I ran a CXTFIT simulation, but please correct me if these inputs seem incorrect:
1. I set up an inverse problem with a deterministic equilibrium CDE.
2. selected time and position as dimensional (cm; days; mg/l)
3. selected flux-averaged concentration, Cf
4. I had "No constraints for parameter estimation" and "No estimates for total mass" and maximum iterations=100.
5. I requested fitted values for v and D and I provided initial estimates of 100 cm/day and 100 cm2/day for them.
6. I selected a "Multiple pulse input" with only 1 pulse:
Pulse 1 (0.2093 mg/litre-cm) with a final time of 0.295 days
*There is perhaps a better way of doing this?? I'm trying to create a pulse of 6.174 mg/l over a duration of 0.295 days, starting on day 0. The water flux is 100 cm/day, hence I divided my solute concentration flux by this value as it says in the manual for flux-averaged concentrations.

The remaining inputs were as follows: zero initial concentraion and no production;
Inverse data structure of Z,T,C and 13 data points.
The data are BTC data taken at a depth of 50cm between time 0.049 and 0.6423 days. I converted my observed concentration data to flux-averaged values (again by dividing through by 100 cm/day).

Lastly, I provided output structure as follows:
7 output positions; 10cm spacing, 0 as my initial position;
800 output times, 0.0125 day for time increment; 0 as my initial output time; and concentration vs time for output code.

==================
The final results indicate that after 19 iterations,
v is 0.7E-4 cm/day and D is 0.168E+4 cm2/day with a MSE of 0.398E-5 and R2 for regression of 0.964.

===================

Would you please correct me if you see any obvious errors in how I set up this problem? I've also attempted a Hydrus simulation, but I'll leave that for another time.

I appreciate any advice.
Thanks,
Elise


Ntoride, 01/10/2008
Elise,
Although it seems to be OK with your input setting, please send me input files (project folder + cxt file). I will have look at the input and output files.
Nobuo


Elise, 01/17/2008
Please find below a record of the e-mail between Nubuo and Elise regarding the request for help on modeling Uranine BTCs. I hope others will benefit for this very helpful advice.
Kind regards,
Elise

-----Original Message-----
From: Nobuo Toride [mailto:ntoride@bio.mie-u.ac.jp]
Sent: Friday, 11 January 2008 10:25 PM
To: Bekele, Elise (CLW, Floreat)
Subject: Re: Inverse modeling Uranine BTCs with CXTFIT

Elise:
Enclosed is a modified project. I corrected the settings for the boundary condition, inverse data, and output data.

>The water flux is 100 cm/day, hence I divided my solute concentration
>flux by this value as it says in the manual for flux-averaged
>concentrations.

As van Genuchten and Parker (1984) discussed, flux conc. is a correction to use a solution for a semi0infinite system for an effluent concentration. Hence you do not need to divide the concentration by the water flux.

> By the way, my initial attempts failed when I provided a full BTC
> dataset (including the descending limb). Is this to be expected? I am
> sending you an Excel file that shows the full BTC obtained for the Red
> Station, but I was only able to get CXTFIT results when I used the
> ascending limb of the BTC.

I could not find a problem in your project.
If any further questions, please do not hesitate to ask me.
Regards,
Nobuo

--------------------------------------------------------
Nobuo Toride, Faculty of Bioresources, Mie University
1577 Kurimamachiya-cho Tsu, Japan, 514-8507
Phone: +81-59-231-9588 Fax: +81-59-231-9604 E-mail ntoride@bio.mie-u.ac.jp
================================================================

-----Original Message-----
From: Nobuo Toride [mailto:ntoride@bio.mie-u.ac.jp]
Sent: Thursday, 17 January 2008 9:26 AM
To: Bekele, Elise (CLW, Floreat)
Subject: Re[2]: Inverse modeling Uranine BTCs with CXTFIT

Elise:

> Would you please tell me if I should be concerned that the model does
> not entirely fit the peak concentration? The overall shape of the
> curve is well-defined, but there are 3 data points that define the
> peak that are not fit by the model.

As described in p.228- in Jury and Horton's soil physics textbook, the CDE uses an analogy of the molecular diffusion for the hydrodynamic dispersion. If the microscopic heterogenous solute transport is different from the assumptions for the CDE, you may observed the irregular transport in the observed BTCs.

The mobile-imobile model is a possible modification, and the stream tube mode assuming no solute mixing in the cross-section area is also a possible candidate for these heterogeneous behaviors. As mentioned before, you also need to consider that you assumed constant v and theta for the CDE fitting.

You can find a lot of studies in soil physics and hydrology literatures regarding the solute dispersion model. It really depends on the situation how much we should take into account for these heterogenous behaviors for the transport modeling.

If any further questions, please do not hesitate to ask me.
Regards,
Nobuo


## ID=914, SDRI inverse solution BC question

Tim01/12/2008
I am using HYDRUS 2D with the inverse solution option to analyze data from a sealed double-ring infiltrometer (SDRI) test. We have no pressure head data, therefore, I have set the initial condition using water content (on the Iteration Criteria dialog). We have flow rate data, which I have input on the Data for Inverse Solution screen. This seems to cause the program to establish the top boundary as specified-flux, but in reality it is constant-head and the observed flux varies as the gradient changes. When I try to specify a constant pressure head value for the top boundary, the program appears to be using the value as water content instead. How do I specify a constant pressure head for the top boundary? I must be missing something obvious.

Jirka,   01/12/2008
Tim,
When the initial condition is given in terms of the water content, then constant head BC is also in terms of water content (and converted in the program into the pressure head) (I know, I should make it clear in the program and I have put it on my list of things to do; sorry for that). This is because the constant head BC values are stored in the same vector as initial conditions.

To specify the constant head BC (with a positive head) in this case (when initial condition is in water contents), please use the time-variable head BC (even with only one value for the duration of the entire simulation). That will work,

Jirka


Tim,   01/12/2008
Thanks Jirka! I did this and still have more questions:

In the Inverse Solution dialog (Flow and Transport Parameters), I specified zero iterations, intending to run a direct simulation and compare results against the observed inflow rates. However, the program took the inflow rates from the Data for Inverse Solution screen and injected them at the top boundary at each specified X (time). Also, in viewing the results, the observed fluxes are now plotted on a different graph than the calculated fluxes, making comparison difficult: the calculated flux appears on the variable boundary graph and the observed flux appears on the constant boundary graph.

Did I misunderstand the way the zero-iteration specification works? What I want to do is run the model with the constant-head top boundary and visually compare the calculated influx against the observed influx, until I get something that looks reasonably close, and then increase the iterations to allow HYDRUS to optimize the parameters. Is this possible?

EDIT: I just realized that I need to change the "Position" specification on the Data for Inverse Solution screen to 3, since the top boundary is now a variable pressure head. That got the fluxes onto a single graph. But the program is still taking the fluxes specified on the Data for Inverse Solution screen, and injecting them through the top boundary. How can I get it to ignore these for a zero-iteration run?

I also have a massive water balance error appearing after 4 days, but I suspect that is related to the handling of the observed flux.


Jirka   01/13/
Tim,
You wrote: "However, the program took the inflow rates from the Data for Inverse Solution screen and injected them at the top boundary at each specified X (time)."
That is not possible.

You wrote: "Also, in viewing the results, the observed fluxes are now plotted on a different graph than the calculated fluxes, making comparison difficult: the calculated flux appears on the variable boundary graph and the observed flux appears on the constant boundary graph."

That is correct as you specified variable flux BC and likely still have Position (in the inverse data) equal to 1 (for constant head or flux). You need to check Position to 3 (which is the code for the variable head BC).

You wrote: "Did I misunderstand the way the zero-iteration specification works?"

You understood it correctly. When you have zero inverse iteration, the code is run in the direct mode and both model predictions and inverse data are displayed. To make such preliminary run, before you actually start doing calibration, is very sensible and I always recommend that. That is because you can see if the system is defined correctly (i.e., you compare the same fluxes, which at this moment you don't. You need to change Position).

You wrote (again): "But the program is still taking the fluxes specified on the Data for Inverse Solution screen, and injecting them through the top boundary. How can I get it to ignore these for a zero-iteration run?"

This is indeed not possible. The direct solver is not aware of what is in the inverse data, except their timing, so that it can send back model predictions.

Jirka

Tim,  01/13/2008

Okay, to see what is really going on, I simplified by unchecking the Inverse Solution option, so am trying to run a simple simulation of infiltration. Unfortunately it is not working and the model is producing some very odd results.

I've set up a rectangular axisymmetric domain 34 cm wide and 61 cm deep, with an 11x31 mesh; vertical node spacing 1:5 top to bottom. No-flow boundaries on each side, a free-drainage bottom BC, and a variable-head BC at the top. The variable head is specified as a constant value of 15 cm throughout the simulation. Initial condition is specified in the water content (0.271 at every node, corresponding to the measured moisture content in the test pad).

When I run this, after a few time steps the program redistributes all the water contents, establishing a gradient from top to bottom and greatly decreasing the water content and pressure head in the lower half of the domain. Then in later time periods the water contents appear to oscillate in the upper part of the domain, like tidal fluctuations!

It seems like part of my problem could be that the test pad is in a nonequilibrium state. Water content is uniform thoughout the vertical profile, because it was constructed that way and there has not been sufficient time for water to equilibrate in the very low-permeability material. HYDRUS seems to want to redistribute the water to create an equilibrium gradient across the vertical dimension.

I thought HYDRUS, with its inverse solution capability, would be an ideal tool for evaluating SDRI data. Can I model this nonequilibrium condition, or do I have to somehow assume a gradient?


Jirka,   01/14/2008
YOu must be doing some strange things. Can you send me the project file (project_name.h3d). I will look at it on Thursday, as I'm traveling till then.

Jirka


Tim,   01/14/2008
I'll be happy to send it to you - thanks very much.
I tried changing the bottom BC to constant head (using another variable-head function) and still got similar results. Very odd results! Can't help thinking it is somehow related to specifying initial conditions in the water content rather than the pressure head.


Jirka,   01/15/2008
Tim,
HYDRUS (2D/3D) is distributed with two examples that demonstrate its use for the inverse analysis of the tension infiltrometer data (one for a homogeneous soil profile and one for a two-layered system). This analyses must be very similar to what you want to achieve. Please, look at these two examples. I'm pretty sure that these two examples will answer most of the questions you have.

Jirka


Tim,   01/15/2008
The examples didn't really clear anything up for me. As far as I can tell I have set up my problem similarly, yet it will not compute properly. So I have been changing one variable at a time to see if I can at least figure out where the difficulty is coming from (mesh, time step, BC, soil parameters...). It appears to be related to the depth of the ponded water, i.e. the head specification in the variable-head BC. Standing water in the SDRI is maintained about 56 cm above the soil surface, so I have been using that as the head condition. If I reduce it to less than 40 cm, I can get the simulation to run, although there is still a relatively large water balance error (12% at the end of the first day, increasing to >40% after 25 days). Above 40 cm, the water balance error apparently becomes too large and causes the program to "flood" the column. I tried ramping up the head incrementally to avoid shockloading, but it still produces the same water balance error as soon as the head gets too high. The more I lower the head, the better the calculations look. At 15 cm the water balance error is around 7%.

Now I'm going to try holding it at 30 cm and changing the mesh, to see if increasing or decreasing the node spacing at the top affects this problem. It seems to be related to both the large head value, and the small Ks value. The examples provided with the

model are for much more permeable soils. How do you typically set up for a low-permeability soil with large ponded head?

Tim,   01/17/2008
After a lot more trials, I have this model running. Basically I reduced the applied head and increased Ks until the model ran without too much water balance error, then used the inverse solution to optimize parameters, then increased the applied head, and iterated that process until I got a decent run with the full 56 cm of applied head. It still has significant water balance error, but at least it produces reasonable-looking results. Not the Ks I expected, or that my client was expecting, but we're getting somewhere.

Jirka,  01/18/2008
Tim,
I have looked at your file and made several changes in the input file. Your problem stemmed from the fact that you are using some really strange material which has a saturated hydraulic conductivity less than 0.02 cm/d. For that you need to use smaller water content and pressure head tolerances and also allow smaller time step. Then I could keep a mass balance error about 2% (which I would still consider large, but for this strange material may be acceptable). I will email you updated file.

Jirka

Tim,  01/19/2008
Thanks Jirka! I think I had mistakenly increased the tolerances, which helped get a solution to a previous problem. Sounds like this time it made things worse instead... I probably need to re-read some sections of the technical manual.

Do you (or anyone else reading this) know of any published work describing inverse modeling of SDRI test data?

Jirka,  01/22/2008
I'm not aware of studies that would actually analyzed axisymetrical flow from ring infiltrometers using inverse modeling. One dimensional analysis was carried out by Russo et al. (1991). I have written several papers on tension disc infiltrometer (which should be rather similar, except that instead of positive pressure head, one has negative heat at the soil surface) (Šimùnek and van Genuchten, 1996, 1997; Šimùnek et al., 1998, 1999; Hopmans et al., 2002). Another interesting study is by Ramos et al. (2006).

Jirka

Russo, D., E. Bresler, U. Shani, and J. C. Parker. 1991. Analysis of infiltration events in relation to determining soil hydraulic properties by inverse problem methodology. Water Resour. Res. 27(6):1361-1373.

Šimùnek, J., and M. Th. van Genuchten, Estimating unsaturated soil hydraulic properties from tension disc infiltrometer data by numerical inversion, Water Resour. Res., 32(9), 2683-2696, 1996.

Šimùnek, J., and M. Th. van Genuchten, Estimating unsaturated soil hydraulic properties from multiple tension disc infiltrometer data, Soil Science, 162(6), 383-398, 1997.

Šimùnek, J., R. Angulo-Jaramillo, M. Schaap, J.-P. Vandervaere, and M. Th. van Genuchten, Using an inverse method to estimate the hydraulic properties of crusted soils from tension disc infiltrometer data, Geoderma, 86(1-2), 61-81, 1998.

Šimùnek, J., D. Wang, P. J. Shouse, and M. Th. van Genuchten, Analysis of a field tension disc infiltrometer experiment by parameter estimation, Intern. Agrophysics, 12, 167-180, 1998.

Šimùnek, J., O. Wendroth, and M. Th. van Genuchten, Estimating unsaturated soil hydraulic properties from laboratory tension disc infiltrometer experiments, Water Resour. Res., 35(10), 2965-2979, 1999.

Hopmans, J. W., J. Šimùnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: Methods of Soil Analysis, Part 1, Physical Methods, Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Ramos, T. B., M. C. Goncalves, J. C. Martins, et al., Estimation of soil hydraulic properties from numerical inversion of tension disk infiltrometer data, Vadose Zone Journal, 5(2), 684-696, 2006.

### ID=915, Do I need Meshgen-2D?

Luna   01/15/2008
Hey,
I am preparing to buy HYDRUS-2D for my Thesis work and I am unclear if I need the stardard version with Meshgen-2D or if I can get away with the cheaper HYDRUS-2D lite.

I am hoping to model unsaturated/saturated flow through a sloping hillslope feeding a stream.

Any help is appreciated
Thanks!


Jirka, 01/15/2008
Luna,
It depends how complicated geometries you want to simulate. If you can describe your transport domain using a simple quadrilateral shape, then you may not need

MESHGEN-2D. Perhaps you should go with that option 2D-Light as you can always upgrade to the full 2D-Standard option.

Jirka


**ID=916, Difference between models**

IJsbrand,  01/15/2008

Hi Jirka or other developers,

I have a question I read an article that there was model of Hydrus-2d developed which was capable of simulating overland flow. It was for modeling Water flow Patters in Flexible Pavements. From Hansson, Lundin en Simunek.
It was concluded that the main problem were the determination and hydraulic properties of the road-materials. I measured some of these materials and determined with pressure plate experiment pF curves.

When I used these parameters it seemed to be more of a problem to obtain a good infiltration pattern in the verge than the horizontal flows in the (sub)-base of the road.

Is it maybe possible to run one of my simulations with the overland flow model? (2d, I presume?) I would love to see if there was a difference with the approach that I used.

Furthermore some of these materials also have an intraparticle flow. As it was only a very low amount compared with the flow around the particles I did not include this in the model.
Is it however (still) possible to include this when using a "pF curve " as table format input?

Greets Ijsbrand de Haan


Jirka,  01/20/2008
Ijsbrand,
I will email you this version of the code together with one example (in few days, as I need to put it together again, having not used it since the paper you refered to was published). But please, be aware that this code was not meant for general distribution and thus is not as stable (that's why it may not work with hydraulic properties given in Tables) as the general HYDRUS (which some may disagree with).

Jirka

IJsbrand, 01/22/2008
Thanks Jirka,
I will incorporate it maybe in my dissertation, if there is a signific difference.

But about the intraparticular flow, I can still put a pf curve ( a strange one with intraparticular flow incorperated) as a table in the orginal hydrus-2d?.

I know I ask a lot of the good ol'model, but thats part of my dissertation to stretch the theory to the boundary and beyond to new materials. Don't  the researcher..

gr.
IJsbrand


**ID=917, Root distribution with non-uniform top surface**

Kgreaser,  01/17/2008
Is it possible to define root distribution over a top surface that is non-uniform in Z using the 'Root Distribution Parameters' window? I'm trying to model a slope in 2D (XZ plane). If I use the window and insert the three parameters (max rooting depth, depth of max intensity and pz) the GUI applies that distribution starting with the highest Z at the top of the slope. But I need the parameters distributed according to the top surface, which gradually declines in Z. Even using the linear distribution with depth option, it still applies it starting with the maximum Z. Bascially, I need depth to be calculated from the top surface, not just Z alone.
Hopefully, a screen shot is included showing the problem.

I would appreciate any suggestions.
Thanks,
Kelly


Jirka,  01/18/2008
Kelly,
At present none of these functions for root distribution takes slope into effect. It is a good suggestion that we should have an option for changing root distribution with slope similarly as we have for initial conditions. We will implement it into the next update, but at present such option is not available.

Jirka


**ID=918, Floating point error – zero divide!**

geo-perre,  01/22/2008
Hello,
in subject line you can see the Error-message I get if I try to run Hydrus 2D with CW2D calculation. Everything else work very well

This occur since this Year, also by Projects wich have done very well before. I didn`t make any changes to the Projects or to the Installation of Hydrus.

If anyone could help or has an idea i would be deeply gratefull.
Thank you
geo-perre


Jirka, 01/22/2008
As i wrote in the other place where you wrote the same, I can send you newly recompiled exe file if you send me an email directly (I can not see your email here).

Jirka


**ID=919, Floating point error – zero divide!**

geo-perre, 01/22/2008
Hello,
in subject line you can see the Error-message I get if I try to run Hydrus 2D with CW2D calculation. Everything else work very well
This occur since this Year, also by Projects wich have done very well before. I didn`t make any changes to the Projects or to the Installation of Hydrus.

If anyone could help or has an idea i would be deeply gratefull.
Thank you
geo-perre


Jirka, 01/22/2008
Geo-Perre,
The CW2D module has never been officially released with the older version of HYDRUS-2D. It was officially released only in March 2006 with HYDRUS (2D/3D) (Langergraber and Šimùnek, 2006). Thus it is difficult to comment on that an official version. I can send you the newly compiled exe if you send me an email directly.

Jirka


Langergraber, G., and J. Šimùnek, The Multi-component Reactive Transport Module CW2D for Constructed Wetlands for the HYDRUS Software Package, Manual – Version 1.0, HYDRUS Software Series 2, Department of Environmental Sciences, University of California Riverside, Riverside, CA, 72 pp., 2006.


geo-perre, 01/22/2008
Hello Jirka,
thank`s for quickley response, I send you a mail, as you requestet.
geo-perre

geo-perre, 01/23/2008
Jirka,

thank you again for quick support, the new exe-File works, it starts with numerical solution but brakes after 30 seconds with:

"Error when reading from an input file Atmosph.in Atmospheric Informations!"

But the Atmosph.in file didn`t change. Perhaps you have an idea?

Im using hydrus-2D version 2.05
with the new CW2D.


Jirka, 01/23/2008
I have not realized that you used old HYDRUS-2D (for which CW2D has never been officially released) as you posted this question in both HYDRUS-2D and HYDRUS (2D/3D) discussion forums. Thus I sent you a compiled file for HYDRUS (2D/3D), which has slightly different formating.

J.


**ID=921, Spatial discretization – convergence**

Gisela, 01/24/2008
Dear everybody,

I am modeling water and solute transport in convection-dominated soil columns with sharp concentrations fronts. In one case, the model often does not converge or shows great oscillations. To avoid this, I tried for example to change the spacial discretization. I do not understand much about the numerics of Hydrus, but I thought a finer discretization should give better results.

Paradoxically, my model gives better results with a coarser discretization: The mass balance errors of the water transport are smaller, the ones of the solute transport are about the same, and the model has no problems with converging. The Peclet-numbers are a little higher but still smaller than 2.

Can this be explained somehow? Or do I missunderstand some principle aspects of the numerics?

I would really be pleased if you could help me.
Gisela

Jirka,  01/24/2008
Gisela,
Your spatial discretization should be such so that the Peclet number is smaller than or equal to 2. There is no need to have a fine grid. There should be no convergence problem with solute transport unless you are using nonlinear adsorption isotherm (Langmuir or Frendlich), since there is no iteration process involved for the linear problems. If you have nonlinear adsorption, then you need to carefully select iteration concentration criteria (the relative tolerance should be about 0.01 to 0.001, absolute tolerance depending on the units you are using).

Jirka


**ID=922, FE mesh sizing**

Brian,  01/24/2008
I am modeling, in 2D, the influx of pre-treated wastewater through 4 variable flux boundaries (BCs) into a variably saturated domain which consists of sandy clay loam (Ksat = 0.177 m/day) overlying sand (Ksat = 9.159 m/day). The units are of relatively equal thickness. Total model length (x) = 275 m and height (z) = 16 m. The center of the model is a mound, with low elevations at each side of the mound which increase in elevation toward the model edges (similar to the shape of the letter "W" with much less topographic relief).

The pressure head initial condition mimics the topography, with the 0 m pressure head surface having a maximum elevation beneath the mound surface, and then descending and then flattening in each direction toward the model edges. The 0 m pressure head surface (water table) is located in the upper sandy clay loam unit only.

There are constant head boundaries (BCs) at the two sides of the model. The pre-treated wastewater is discharged to the subsurface through four variable flux BCs located at the top of the mound, which are also characterized by varying values over a 214 day time period (range = 0 to 0.063 m/day). I have also used seepage face BCs at the two lengths of low topographic elevation on either side of the mound, and atmospheric BCs at the remainder of the land surface. The atmospheric BCs are characterized by 214 days of precipitation data (range = 0 to 0.049 m/day).

I have created the model three times, using different mesh (Targeted FE) sizes:
1. 1 m
2. 0.3 m
3. 1 m, with 0.3 m mesh refinement at interface between sandy clay loam and sand units, and at variable flux BCs at top of mound.

When I run any of these models, the "ORTHOMIN TERMINATES – too many iterations, time step reduced" phrase is printed immediately or within only a few (very short) time steps.

I suspect that my mesh may be too coarse for all three models (at some critical locations), but I am not sure: is there a rule of thumb for FE mesh coarseness in this situation? I have tried altering pressure head and water tolerances, and initial and

minimum timesteps. (I am using the van Genuchten-Mualem hydraulic model with -2 cm air entry value.) I have also conducted some model experiments with the pressure head surface instead having a small gradient from the left to the right side of the model (i.e., no mounded water table at time = 0); in this case, the model will run for a few iterations, say to time = 0.003 days, but then revert to the ORTHOMIN TERMINATES phrase.

I have been reading through the manuals and searching the Discussion Forum, which has certainly been helpful as I research this issue, but as of now I am stuck. Any suggestions and/or advice would be greatly appreciated!

Brian


Jirka, 01/25/2008
Brian,

You have a relatively large transport domain (275*16 m). If you use Targeted FE size of about 0.3, you will end up with a huge number of finite elements (about 150,000). While I would recommend having vertical discretization even finer in the unsaturated zone (30 cm in the saturated zone would be just OK), you can have much larger elements in the horizontal direction. You can achieve that by using anisotropic mesh. Although you may find in the literature that elements should be similar in different direction, they should correspond more with the direction of pressure head gradients. As the pressure head gradients are largest in the vertical direction you need to have finer discretization in this direction. Thus I would recommend trying the mesh anisotropy of up to 1:10. If you use targeted FE size of 10 cm and a stretching factor of 10 (FE-Mesh Parameters, Tab Smoothing) would lead to about 150,000 elements (the same as above, but better grid).

The other thing that you may try is to use smoothing factor larger than the default value of 1.3. If you use the smoothing factor (FE-Mesh Parameters, Tab Options 1) of 3, then you will have about 35,000 elements.

Be also careful how you specify the initial condition. If you specify saturated parts in different parts, you may create unreasonable gradients that may cause these problems.

Jirka


Brian, 01/25/2008
Hello Jirka,

Thanks much for the advice. I have implemented the changes as described and generated an anisotropic mesh at 1:10 and increased the smoothing factor to 3, as well as changed the soil hydraulic model to the regular van Genuchten-Mualem model. Results are described as follows:

For the model with the pressure head initial condition having a water table mound, the ORTHOMIN TERMINATES – too many iterations, time step reduced phrase is printed to screen immediately and repeatedly.

However, for a second model run where I created a pressure head initial condition with hydraulic gradient = 0 and saturated thickness of a 5.5 m, the model ran quite well.

The problem with the first model must be in how I am specifying the initial condition: by creating a water table mound as an initial condition, I must be causing unreasonable gradients. I am selecting the initial condition to create 4 zones, from left to right in the model, as described below with pressure head = 0 m (x, z) coordinates:
1. Gradient = 0, coordinates: (0.0. 341.02) to (51.25, 341.02)
2. Gradient = 0.0311 (1.78 degrees), coordinates: (51.25, 341.02) to (111.66, 342.25)
3. Gradient = -0.0311 (-1.78 degrees), coordinates: (111.66, 342.25) to (214.14, 340.33)
4. Gradient = 0, coordinates: (214.14, 340.33) to (273.89, 340.33)

I used the on-screen pick-box to select each of the four initial condition regions, using equilibrium with lowest located nodal point with the appropriate value at the lowest located nodal point and corresponding theta (gradient) value for each region. I understand that the peak and valleys created in this initial condition are not realistic in that the sharp changes in gradient are not truly representative of the water table. Is there a better way to represent this? Could I somehow create a smoothed input file for initial condition?

I also conducted a third model run where I eliminated items 1. and 4. from the initial condition, so that a "peaked" water table with no flattening toward the model edges was created. This model also presented me with the ORTHOMIN TERMINATES result.

Perhaps there is some way to smooth the gradient that I am unaware of?

Thanks much.
Brian

## ID=923, Atmospheric boundary condition with rainfall

Eessig, 01/28/2008
I have a question related to the atmospheric boundary conditions with rainfall. I have a box of soil approx. 1.5 M long, 1.2 M wide, and .7 M deep. No flux on the sides, free drain on the bottom, and atmospheric on the top; initial conditions based on water content. 15 hr experiment, rainfall applied for the first 8 hrs, no rainfall afterward. How does the atmospheric boundary conditions handle the calculation after reaching saturated soil conditions? Our calculation runs fine until saturated conditions, then takes the maximum amount of iterations to converge, and then the hAtm results stop making sense (jumping, then going extremely negative). What do you think is going wrong?

Jirka, 01/28/2008

When the system is fully saturated and boundary conditions are flux at the surface (e.g., atmospheric BC) and free drainage at the bottom, then the problem has an infinite number of solutions since the system can carry this flux at any positive pressures (q=-K gradH). The code can not solve such problem. Boundary conditions has to be specified in such a way so that there is a unique solution.

Jirka

Eessig, 01/29/2008
Thanks for your response.

With the same soil column, I changed the bottom boundary to a constant water content (=saturation implying atmospheric pressure). The top boundary is an atmospheric boundary condition with a prescribed rainfall. The initial condition is close to saturation everywhere. It still does not work, and the surface pressures start to fluctuate after a few minutes.

Previously, under the atmospheric boundary condition at the surface and free drainage at the bottom, the program seemed to run fine initially, but gave odd results as soon as the rain stopped. The water front did not reach the bottom, so I am not sure if the bottom boundary condition was engaged.

Actually, the top boundary condition should be a seepage face because surface flow was collected from the soil surface. But when we used the seepage face boundary condition option, it did not seem to read the rainfall data at all. Does the program not allow rainfall on a seepage face?

In the physical experiment, deep flow was also collected. I thought a free drainage boundary condition would allow me to simulate this hydrograph as well. What would be a more appropriate lower condition?

Jirka, 02/04/2008
Eesig,

If the moisture front did not reach the bottom of the soil profile, then the combination of the atmospheric upper BC and free drainage bottom BC should be OK, and the code should work fine when proper spatial and temporal discretization is used.

Surface boundary should certainly not be a seepage face (that is a boundary for an outflow from the soil profile, such as bottom of lysimeters, down face of a dike, etc). The code allows rain on inactive part of the seepage face, but you need to select that option in the BDRC options.

If you have soils with low n (lower than 1.2) then you may want to use the model with the -2 cm air-entry value.

Jirka

**ID=925, Difference between sum (infil.) and precipitation**

yxb967,  01/30/2008
I'm modeling winter wheat evapotranspiration from a shallow groundwater table.
I have got the parameters of the soil which is divided into two sections in inverse problem . As follows:

Variable Value S.E.Coeff. Lower Upper
WCR 6.28E-02 4.48E-01 -8.37E-01 9.62E-01
WCS 3.69E-01 4.00E-02 2.89E-01 4.50E-01
ALPHA 9.63E-03 2.17E-02 -3.40E-02 5.33E-02
N 1.12E+00 3.54E-01 4.11E-01 1.83E+00
CONDS 2.30E+00 1.68E+01 -3.14E+01 3.60E+01
L 3.22E-01 8.49E+00 -1.67E+01 1.74E+01

WCR(not fitted) 0.075
WCS 4.28E-01 3.08E-02 3.66E-01 4.90E-01
ALPHA 7.65E-03 2.07E-02 -3.40E-02 4.93E-02
N 1.30E+00 9.76E-01 -6.56E-01 3.27E+00
CONDS 1.79E+01 1.05E+02 -1.94E+02 2.30E+02
L 2.72E-02 2.60E-01 -4.96E-01 5.51E-01

The parameters are reasonable?

I modeled wheat ET with precipitation using the parameters above in 237 days but the results is puzzling.The difference between the sum(infil.) and precipitation without runoff is large.

Spatial discretization: about 1 cm
Water content tolerance: 0.001
PRessure head tolerance: 1 cm
Initial time step: 0.0001 d
Minimum allowed time step: 1e-7 d
Max time step 0.5

who can help me?
thank you
yxb


Jirka,  01/30/2008
YOu do not give units and thus it is difficult to judge. I assume you have cm and d.

If you have precipitations larger than CondS of the upper layer (2.3 cm/d) then it is likely that you will have surface run off and there will be difference between cum(prec) and cum(infil). The difference should be in the Runoff. Check the mass balance error. If it is below 1% then calculations are likely correct and you should be able to figure out the water balance (it should be closed).

Jirka

yxb967, 01/31/2008
Sorry,I forgot. The units are cm and d.
The Max. of precipitation is 7.5cm larger than 2.3cm/d but I have set Max h at soil surface 8cm/d.So there will not be surface runoff.
As to the mass balance errors,they range from 12.445% ~ 1.193% and decrease with time in a 237-days simulation period.I don't undstand why the errors decrease with time.Is it because of the impact of initial condition£¿
How should I improve the precision?

thank you


Jirka, 01/31/2008
Probably use finer spatial and temporal discretization (allow the minimum time step to be on the order of 1s). You can also use more strickt iteration criteria (for both water content and pressure head) than dafault values.

J.


Dehy, 02/01/2008
Hi Jirka,
In our simulation, the ponding depth is assumed 8cm, which is larger more than the irrigation water depth and most of the single rainfall event water depth. The simulation results showed that runoff is almost zero, which is reasonable under the assumption. However, the infiltration for each irrigation application is always much less than the water applied, the discrepancy of which is 3-4cm or so. For a infiltration event of 3 days or so, evaporation is negligible. We can not figure out why the water was 'lost'.

1. Are there any other possible paths of water flow? What is the problem with this simulation?
2. Any suggestions to this problem?

Your help will be highly appreciated.rainfall


## ID=927, Rainfall-infiltration disparity

Dehy, 02/03/2008
Although quite several posts have discussed the disparity of rainfall and precipitation, i still cannot figure out why the disparity is so significant in my case. So, I give the detailed description to my simulation and results as following. Your comments to this problem will be aprreciated.

Description to the simulation
Soils. two soils. Soil type 1: soil layer 0-30cm. Soil type 2: below the 30cm. Soil model: VG with air entry value -2 cm.
Depth to water: 100 cm

Discretization of soil column: 0.5 cm for each layer.
Upper BC: atmospheric with surface layer. The max h at soil surface: 10cm
Lower BC: variable pressure head. GWL=0
Time variable Boundary conditions: daily precipitation, Evap, Transpi. hCritA = 15000.
Time control and interartion control were set as default.

Simulation results
Total precipitation: 36.61 cm (including 2 irrigations, 7.5 cm for each).
The sum of infiltration: 25.48 cm
The sum of potential evapotranspiration during for the surface ponding days: 6.64 cm
Residue of balance: 36.61-25.48-6.64 = 2.50 cm.

Questions
Where does this portion of water go? What are the possible reasons for the disparity?


**ID=928, Eccentric behabiour of capillary rise**

Dehy,  02/03/2008
There are three rainfall/irrigation events in my simulation. Something that made me confusing happened for the first event, an irrigation of 75mm water in depth. The capillary rise did not turned to percolation that should have happened because of the irrigation input. Instead, the capillary rise increased sharply during this day! For other two sequential rainfall/irrigation events, the simulated capillary rise and percolation was ok.

What is the matter with it?
Any similar experiences or comments? Thanks in advance.
Dehy


**ID=929, V-G parameters**

BoyC,  02/06/2008
Hello all,
Anyone out there has Van Genuchten parameters for a (karstic) limestone? It would be greatly appreciated if you can post them. Thanks!

BoyC


**ID=930, Nonequilibrium**

Srilert, 02/08/2008
Dear all,
I would like to ask about the procedure for estimating parameters using HYDRUS1D for physical nonequilibrium and chemical nonequilibrium with my experimental results.

If in my experiementalresults, they compose of 2 parts, first is nonsorbing solute transport and second sorbing solute transport.
Could you please suggests the procedure for running the models?
1. Firstly, I fitted the tracer results with physical nonequilibrium models using inverse solution to estimate Fract.(fraction of mobile) and alpha (rate of diffusion) in HYDRUS 1D, then I will get D (dispersivity), èm, and vm .
2. Then, I use D as a effective dispersivity from previous part I as input parameter, and I fitted sorbing solute results with nonequlibrium models using inverse solution to estimate Fract. and Alpha in Hydrus 1D, then I will get fc (fraction of instantaneous sorption site), ác (first-order kinetic rate coefficient), and sorption parameters (k or/ and beta).
Is it right? Please give me any comments


Jirka, 02/08/2008
No, it is not. The reason is that current version can not consider simultaneously both physical and chemical nonequilibrium. It can consider either physical (mobile-immobile water contents model) or chemical (two-site sorption model) nonequilibrium, but not both. If you can wait we will post a beta version of the version 4.x of HYDRUS-1D within a week or two, which will have this option, i.e., to consider physical and chemical nonequilibrium simultaneously.

Jirka



**ID=931, Physical nonequilibrium**

Srilert,02/08/2008
Dear all,
I have read in some papers they refer to dual-posity, dual-permeability models(DPM) or triple-porosity (DP-MIM). but I don't know these models are in the update version of HYDRUS 1D. because I found three of Dual porosity/ Dual permeability models
1.Dual-porosity (Durner, dual van Genuchten-Mualem)
2.Dual-porosity (mobile-immobile, water c.mass transfer)
3.Dual-porosity (mobile-immobile, head mass trasfer)

Could anyone please explain me more details ?

Thanks in advance.
Srilert


Jirka, 02/08/2008
As I wrote in another answer at this site, we will be releasing a beta version of HYDRUS-1D, which will have these options within a week or two. These options are not available in current version (3.x) of HYDRUS-1D.

Jirka