=== Version 2.05 ==========

==========
Version 2.05.0250 – March 3, 2017
==========

Fixed Errors:

1. Deleting materials in “Table of materials” could cause program crash under certain circumstances.
2. Error in the GUI in specifying osmotic coefficients for the stress response function for HP2
3. Minor errors in the export files for ParSWMS (a section of Grid.in was written only when solute transport was enabled; the Theta_k and Theta_m parameters were switched.

==========
Version 2.05.0230 – November 19, 2016
==========

Fixed Errors:

1. Incorrect colors of materials for more than 20 materials.
2. Error in the GUI in calculating the root distribution function Beta.

==========
Version 2.05.0230 – November 19, 2016
==========

Fixed Errors:

1. Incorrect colors of materials for more than 20 materials
2. Error in the GUI in calculating the root distribution function Beta

==========
Version 2.05.0190 – April 19, 2016
==========

New Features:

1. New add-on module SLOPE Cube (Slope Stability)

Fixed Errors:

1. Changes of auxiliary objects (dimensions, comments, etc.) lead to deleting results, which actually was not necessary.
2. Units in the field capacity calculations

=== Version 2.04 ==========

==========
Version 2.04.0670 – April 14, 2016

==========

Fixed Errors:

1. FE-mesh generator – 3D mesh generation could fail when using internal points/lines of Solids
2. Internal points/lines/openings of surfaces were disabled/unusable for curved (non-planar) surfaces
3. FE-mesh generator – 3D mesh generation could fail if a boundary of a Solid was defined by a very large number of surfaces (> 1000)
4. Chart - error in the graphical display of results at observation nodes (for Nobs > 20) observation nodes)
5. Chart – error in the graphical display of a quantity along a mesh-line
6. Fixed tool for graphical input of connected segments
7. Fixed a silent error when saving large projects (size > 2GB)

==========
Version 2.04.0520 – February 15, 2015
==========

Fixed Errors:

1. Small bug in the FE-mesh generator (mesh generation failed in tutorial Demo_501_01)
2. Error in function “Generate boundary lines from points”. This error could subsequently cause a mesh generation failure.
3. Error when importing multiple solutes using function “Import quantity at scattered points”.
4. Improved the stability of “View All” function on computers with Intel mobile graphics drivers (usually notebooks).

==========
Version 2.04.0390 – September 19, 2014
==========

New features and improvements:

1. New HYPAR module: HYPAR is a parallelized version of the standard two-dimensional and three-dimensional HYDRUS computational modules.
2. New SLOPE module: The module analyses the stability of generally layered two-dimensional soil slopes including the presence of water modelled using the pore pressure automatically imported from HYDRUS results.
3. The on-line deactivation no longer requires a password.
4. HYDRUS can automatically check for new updates (using pc-progress.com website) and informs the user if an update is available.

Fixed Errors:

5. Boundary surfaces of internal solids (solids in solids) were incorrectly included in domain boundaries (boundary conditions).
6. Several more minor errors.

=== Version 2.03 ==========

==========
Version 2.03.0600 – April 10, 2014
==========
Fixed Errors:

1. Function "Simplify lines by removing excess points" (dialog "Repair Geometry") used in projects with 3D-Layered domain type could result in incorrectly defined thickness vectors.

==========
Version 2.03.0590 – March 27, 2014
==========
Fixed Errors

1. Network HASP and very long simulations: After finishing a long simulation (several days), the program didn't allow the user to save data because of a lost authorization (NetHASP session has expired).

==========
Version 2.03.0580 – February 11, 2014
==========
Fixed Errors:

1. After finishing the calculation, the calculation module was waiting for the "Enter" key even if the "Press Enter at the End" checkbox was unchecked in the Output Information dialog.

==========
Version 2.03.0560 – January 1, 2014
==========
Fixed Errors:

1. UNSATCHEM Module: fixed a check of solution compositions, which could sometimes stop the calculation.

==========
Version 2.03.0520 – December 9, 2013
==========
Fixed Errors:

1. Fixed Vrugt (2001) root horizontal distribution function (with radius of max intensity).
2. New options for the import of values from scattered points
3. Fixed an error in 3D-Layered domain types that could cause a constant domain thickness instead of variable thickness (defined by a large number of thickness vectors).
4. Fixed: After opening the "Domain Type" dialog, the project was marked as "Modified", although nothing has been changed.

==========
Version 2.03.0450 – October 16, 2013
==========

Fixed Errors:

1. Error in the generation of FE-mesh (3D).
2. Error in the 2D calculation module (both direct and inverse solution). Writing information about special boundary condition options into un-opened file. The problem occurred only when HYDRUS was not run in the administrative mode and when special BC options used.
3. Error during automatic generation of surfaces from lines.

New features and improvements:

1. Extended options for automatic corrections of the geometry model and detection of errors.

==========

Version 2.03.0350 – September 24, 2013

==========

Fixed Errors:

1. Error during the on-line activation of HYDRUS caused by a double-click on the “Activate Now” button.
2. Error when entering multiple comments (graphical objects).
3. DualPerm Module: "Fraction of boundary Flux into Fracture" parameter was not transferred from the dialog window to input data.
4. Incorrect graphical display of the FE-mesh after applying the function “Remove selected elements”.

New features and improvements:

1. Using new fonts compatible with the Chinese, Japanese and other similar Windows systems. This should prevent cutting off text in dialogs and other parts of the GUI.
2. Export of isolines (available only in 3D-Professional).
3. Import if various quantities (pressure head initial condition, etc.) from values defined at scattered 2D/3D points.
4. Allowing triggered irrigation in the UnsatChem module.

=== Version 2.02 ==========

==========

Version 2.02.0720 – August 31, 2013

==========

Fixed Errors:

1. HYDRUS activation: The default expiration date for time-limited activation, which should be 6 month from the current date, was sometimes incorrect (was equal to the current date).
2. HYDRUS activation: It was not possible to set the expiration date after July 10, 2014.

==========

Version 2.02.0700 – February 24, 2013

==========

Fixed Errors:

1. Incorrect error message “Inverse solution/Dual-permeability is implemented only for one solute!” in the Solute Transport dialog. This occurred in DualPerm Module and Inverse solution.
2. Active solute root uptake was inactive in Version 2.01.
3. Porosity in the soil catalog for the Brooks and Corey model was updated based on Rawls et al (1982) (originally this value was based on the RETC report (van Genuchten et al., 1991)).

New features and improvements:

1. Initial Conditions for water flow can be set equal to Field Capacity (Twarakavi et al., 2009).
2. Display of wetting hydraulic functions for hysteretic soils.
3. The maximum time step in the Wetland module is limited to avoid concentration overshoots.

==========
Version 2.02.0660 - September 19, 2012
==========

Fixed Errors:

1. Transfer of anisotropy information from Geo-Objects to FE Mesh.
2. Saving of the iN_BM parameter in the CW2D module.
3. Transfer of soil hydraulic parameters from the Rosetta module.
4. Transfer of 2D domain type (horizontal, vertical, axisymmetric) to the calculation module.
5. Import of initial conditions into 2D-XY domains.
6. Graphical tool for insertion of Quad Surfaces.
7. Import of geometry from text file, block OBJECT=THICKNESS_ARR3Z_NLAYERS
8. Error in evaluating the sizes of boundary lengths/areas that could occur in 3D-general domains composed of multiple solids (more than 1). This could have led to erroneous boundary fluxes for flux BCs.
9. Fixed an incorrect number of time layers in the dialog for import of initial conditions from another HYDRUS project.
10. Error in importing data from another HYDRUS project with different units.
11. Error in the on-line activation under Proxy server.
12. Error in accessing ChemData files from the UnsatChem calculation module (under special circumstances).
13. Error in graphical display of 3D anisotropy on FE-mesh.
14. Fixed several errors in the FE-mesh generator.
15. Error in automatic generation of surfaces from selected lines.
16. Fixed the formatting of some floating-point values in the GUI (BDRC - constant flux and deep drainage, nodal recharge).
17. Error in time units used for boundary conditions defined on geometrical objects.
18. Error in the import of old *.h2d projects (Hydrus-2D & Meshgen-2D).

New features and improvements:

1. Added new function: File -> Import and Export -> Export FE-Mesh. This function exports FE-mesh to a text file that includes information about links between geometrical objects and corresponding FE-mesh elements.
2. Added new function: File -> Import and Export -> Export Current Quantity. This function exports values of the current quantity (displayed in the active view) into a text file. The function exports values only for visible nodes (one can use FE-Mesh Sections to export values for any part of the domain).
3. Updated UnsatChem Module.
4. The new DualPerm module simulating flow and transport in dual-permeability porous media.
5. The new C-Ride module simulating colloid transport and colloid-facilitated solute transport.
6. The new UnsatchemDual module simulating transport of major ions in dual-permeability porous media.
7. Improvement: if there is a difference between values defined on GeoObj and FE-mesh, the program displays a yellow arrow pointing to the first difference.
8. GUI look has been improved by using true-color icons and advanced docking-panel layout.
9. Improvement: Defining objects on geometric objects is possible even after removing some FE-elements from the mesh. Values are then transferred from GeoObj to remaining mesh nodes or elements.
10. Added new option: Authorization of HYDRUS using hardware key (HASP). Starting with version 2.02, the software key (activation) can be used to authorize only separate computers, while a Hardware Key is required for the network installation.

=== Version 2.01 ==========

==========
Patch 2.01.1240 - August 24, 2012
==========

1. Fixed an error in the import of old *.h2d projects (Hydrus-2D & Meshgen-2D).

==========
Patch 2.01.1230 - April 17, 2012
==========

1. Values of the Constant Flux boundary condition could be nullified by specifying a Solute Transport BDRC. This error first appeared in version 2.01.1220.

==========
Patch 2.01.1220 - March 28, 2012
==========

1. Fixed formatting of some floating-point values in the GUI (BDRC - constant flux and deep drainage, nodal recharge).
2. Fixed an error in time units used for boundary conditions defined on geometrical objects.
3. Improvement: if there is a difference between properties or values defined on geometrical objects and FE-mesh, the program displays a yellow arrow pointing to the first difference.

==========
Patch 2.01.1200 - March 18, 2012
==========

1. Fixed several minor errors in the FE-mesh generator.
2. Fixed an error in the automatic generation of surfaces from selected lines.

==========
Patch 2.01.1070 - March 4, 2012
==========

1. Fixed the installation program. The required execution level changed to "Administrator".

==========
Patch 2.01.1060 - February 16, 2012
==========

1. Fixed an error in the on-line activation under Proxy server.
2. Fixed an error in accessing ChemData files from the Unsatchem calculation module (under special circumstances).
3. Fixed an error in the graphical display of 3D anisotropy on FE-mesh.

==========
Patch 2.01.1090 - October 17, 2011
==========

1. Fixed an incorrect number of time layers in the dialog for import of initial conditions from another HYDRUS project.

2. Fixed an error in importing data from another HYDRUS project with different units.

==========
Patch 2.01.1080 - August 28, 2011
==========

1. Fixed an error in evaluating the sizes of boundary lengths/areas that could occur in 3D-general domains.
1. Composed of multiple solids (more than 1). This could have led to erroneous boundary fluxes for flux BCs.
2. Fixed several other minor errors
3. Added new function: File -> Import and Export -> Export FE-Mesh…
4. This function exports FE-mesh to a text file that includes information about links between geometrical objects and corresponding FE-mesh elements.
3. Added new function: File -> Import and Export -> Export Current Quantity…
5. This function exports values of the current quantity (displayed in the active view) into a text file. The function exports values only for visible nodes (one can use FE-Mesh Sections to export values for any part of the domain).

==========
Patch 2.01.1070 - August 08, 2011
==========

1. Fixed an error in the import of initial conditions into 2D-XY domains.
2. Fixed an error in the graphical tool for insertion of Quad Surfaces.
3. Fixed an error in the import of geometry from text file, block OBJECT=THICKNESS_ARR3Z_NLAYERS.

==========
Patch 2.01.1060 - July 03, 2011
==========

1. Fixed an error in the transfer of 2D domain type (horizontal, vertical, axisymmetric) to the calculation module.

==========
Patch 2.01.1050 - June 28, 2011
==========

1. Fixed an error in the transfer of soil hydraulic parameters from the Rosetta module.

==========
Patch 2.01.1040 - June 17, 2011
==========

1. Fixed an error in the GUI for the WETLAND module when using the "old" CW2D model: in the window "Constr.Wetland Par.II" the parameter value "N content of biomass" was not transferred to the calculation correctly.

==========

Patch 2.01.1030 - June 6, 2011

==========

1. Fixed the transfer of properties from geo-objects to FE-elements - see error http://www.pc-progress.cz/_forum/topic.asp?TOPIC_ID=1720.

==========

Version 2.01.1000 - May 6, 2011

==========

New Features and Changes (related to GUI):

1. Supports for complex general three-dimensional geometries (Professional Level).
2. Domain Properties, Initial Conditions, and Boundary Conditions can be specified on Geometrical Objects (defining the transport domain) rather than on the finite element mesh.
3. Import of initial conditions from existing HYDRUS projects even with (slightly) different geometry or FE mesh.
4. Import of various quantities (e.g., domain properties, initial and boundary conditions) from another HYDRUS projects even with (slightly) different geometry or FE mesh.
5. Support of ParSWMS (a parallelized version of SWMS_3D).
6. Support of UNSATCHEM (a module simulating transport of and reactions between major ions).
7. The Mass Balance (Inverse) Information dialog window enables to display texts larger than the capacity of the Edit window.
8. Constructed Wetland Parameters commands added to the main menu and navigation tree.
9. Root distribution can be specified using GUI parallel with the slope for hillslopes.
10. Import: Geometric objects can be imported using DXF and TIN (triangular irregular network) files.
11. Display of results using Isosurfaces.
12. Support of a new CWM1 constructed wetland module.

New Features and Changes (related to computational modules):

1. New more efficient algorithm for particle tracking. Time-step control to guarantee smooth particle paths.
2. Initial conditions can be specified in the total solute mass (previously only liquid phase concentrations were allowed).
3. Initial equilibration of nonequilibrium solute phases with equilibrium solute phase (given in initial conditions).
4. Gradient Boundary Conditions (users can specify other than unit (free drainage) gradients boundary conditions).
5. A subsurface drip boundary condition (with a drip characteristic function reducing irrigation flux based on the back pressure).
6. A surface drip boundary condition with dynamic wetting radius.
7. A seepage face boundary condition with a specified pressure head.
8. Triggered Irrigation - irrigation is triggered by the program when the pressure head at a particular observation nodes drops below a specified value.

9. Time-variable internal pressure head or flux nodal sinks/sources (previously only constant internal sinks/sources).
10. Fluxes across meshlines in the computational module for multiple solutes (previously only for one solute).
11. HYDRUS calculates and reports surface runoff, evaporation and infiltration fluxes for the atmospheric boundary.
12. Water content dependence of solute reactions parameters using the Walker's [1974] formula was implemented.
13. An option to consider root solute uptake, including both passive and active uptake [Šimunek and Hopmans, 2009].
14. The Per Moldrup's tortuosity models [Moldrup et al., 1997, 2000] were implemented as an alternative to the Millington and Quirk [1960] model.
15. An option to use a set of Boundary Condition records multiple times.
16. Executable programs are about 1.5 - 3 times faster than in the standard version due to the loop vectorization.
17. A new CWM1 constructed wetland module (in addition to the CW2D module).
18. New options related to the fumigant transport (e.g., removal of tarp, temperature dependent tarp properties, additional injection of fumigant).

Fixed Errors:

1. Fixed error: The Wetland module had a wrong format statement when writing the CumQ.out file.
2. Fixed error: FE-mesh generation could fail if stretching factors were >1 and the domain boundary contained polylines and (at the same time) splines or arcs.
3. Fixed error: Activation energy coefficients in the temperature dependence functions were incorrectly converted when time units changed.
4. Fixed error: Unit conversion of the area associated with transpiration was incorrected when length units chanded.
5. Fixed error: Conversion of some first- and zero-order rate constants with respect to length units.

Support of ParSWMS

Three-dimensional applications often require a large number of finite elements to discretize realistically large transport domains. Even with the fast personal computers currently available, it is virtually impossible to solve within a reasonable computational time problems having more than about half a million nodes or more. To decrease the required computational time, Hardelauf et al. (2007) parallelized SWMS_3D to develop ParSWMS that distributes problems with a large number of elements over multiple processors working in parallel. SWMS_3D is the simplified predecessor of the 1.0 version of HYDRUS (2D/3D). While SWMS_3D simulates water flow and solute transport in three-dimensional domains, it does not consider some of the advanced features of HYDRUS, such as dual-porosity systems, hysteresis, and nonlinear and nonequilbrium solute transport. The ParSWMS code was developed for the LINUX or UNIX workstations using the installed free-wares MPI, PETSc and PARMETIS. Hardelauf et al. (2007) demonstrated that doubling the number of processors may decrease the computational time by up to nearly 50%.

An extended version of HYDRUS GUI supports fully ParSWMS. It allows users to create the three-dimensional flow and transport project in HYDRUS GUI and then save it using the format of ParSWMS input files. These input files can then be taken to a parallelized platform (a supercomputer or a cluster of PCs), on which ParSWMS can be run. Created output files can then be copied back to a PC with HYDRUS GUI, which will convert ParSWMS-created output files into the HYDRUS format. The results can then be analysed in HYDRUS GUI using all its graphical tools and confort.

Šimůnek, J., K. Huang, and M. Th. van Genuchten. 1995. The SWMS_3D code for simulating water flow and solute transport in three-dimensional variably saturated media. Version 1.0, Research Report No. 139, U.S. Salinity Laboratory, USDA, ARS, Riverside, California, 155 pp

Hardelauf, H., M. Javaux, M. Herbst, S. Gottschalk, R. Kasteel, J. Vanderborght, and H. Vereecken. 2007. PARSWMS: a parallelized model for simulating 3-D water flow and solute transport in variably saturated soils. V Zone Journal, 6(2):255-259.

Support of UNSATCHEM

The geochemical UNSATCHEM module (Simunek and Suarez, 1994; Šimůnek et al., 1996) has been implemented into the two-dimensional computational module of the HYDRUS (2D/3D) software package. The geochemical UNSATCHM module simulates the transport of major ions in variably-saturated porous media, including major ion equilibrium and kinetic non-equilibrium chemistry. The resulting code is intended for prediction of major ion chemistry and water and solute fluxes in soils during transient flow. The major variables of the chemical system in UNSATCHEM-2D are Ca, Mg, Na, K, SO4, Cl, NO3, H4SiO4, alkalinity, and CO2. The model accounts for various equilibrium chemical reactions between these components, such as complexation, cation exchange and precipitation-dissolution. For the precipitation-dissolution of calcite and dissolution of dolomite, either equilibrium or multicomponent kinetic expressions can be used, which includes both forward and back reactions. Other dissolution-precipitation reactions considered include gypsum, hydromagnesite, nesquehonite, and sepiolite. Since the ionic strength of soil solutions can vary considerably in time and space and

often reach high values, both the modified Debye-Hückel and the Pitzer expressions are incorporated into the model to calculate single ion activities. The effect of solution chemistry on the hydraulic conductivity is also considered. Water flow and heat transport modules are similar (almost identical) as in regular HYDRUS. Application of the UNSATCHEM module is demonstrated on several examples.

Šimůnek, J., and D. L. Suarez, Two-dimensional transport model for variably saturated porous media with major ion chemistry, Water Resour. Res., 30(4), 1115-1133, 1994.

Šimůnek, J., D. L. Suarez, and M. Šejna, The UNSATCHEM software package for simulating one-dimensional variably saturated water flow, heat transport, carbon dioxide production and transport, and multicomponent solute transport with major ion equilibrium and kinetic chemistry, Version 2.0, Research Report No. 141, U.S. Salinity Laboratory, USDA, ARS, Riverside, California, 186 pp., 1996.

Walker, A., A simulation model for prediction of herbicide persistence, J. Environ. Quality, 3(4), 396-401, 1974.