Notes on Spatial and Temporal Discretization
(when working with HYDRUS)

Jirka Šimůnek and Miroslav Šejna

This text was last modified on December 27, 2024.

1. Temporal Discretization

Four different **time discretizations** are used in HYDRUS:

a. time discretizations associated with the numerical solution,
b. time discretizations associated with the implementation of boundary conditions,
c. time discretizations, which provide a printed output of the simulation results (e.g., nodal values of dependent variables, water, solute mass balance components, and other information about the flow regime), and
d. time discretizations associated with data defining the objective function in the HYDRUS inverse mode (e.g., measured water contents, pressure heads, concentrations, and/or fluxes).

Discretizations 2, 3, and 4 are mutually independent; they generally involve variable time steps as described in the input data file. Discretization 1 starts with a prescribed initial time increment, Δt_{init}. The time increment, Δt, is automatically adjusted at each time level according to the following rules:

a. Discretization 1 must coincide with time values resulting from time discretizations 2, 3, and 4.
b. Time increments cannot become less than a preselected minimum time step, Δt_{min}, nor exceed a maximum time step, Δt_{max} (i.e., $\Delta t_{min} \leq \Delta t \leq \Delta t_{max}$).
c. If, during a particular time step, the number of iterations necessary to reach convergence is ≤ 3 (It_{min}), the time increment for the next time step is increased by multiplying Δt by a predetermined constant >1 (k_1, usually between 1.1 and 1.5). If the number of iterations is ≥ 7 (It_{max}), Δt for the next time level is multiplied by a constant <1 (k_2, usually between 0.3 and 0.9).
d. If, during a particular time step, the number of iterations at any time level becomes greater than a prescribed maximum (It_{crit}, usually between 10 and 50), the iterative process for that time level is terminated. The time step is subsequently reset to $\Delta t/3$, and the iterative process is restarted.

Parameters Δt_{init}, Δt_{min}, Δt_{max}, It_{crit}, It_{min}, It_{max}, k_1, and k_2 are specified by a user at the input. The recommended values for these parameters are as follows:

Parameter	Recommended Value	Comment
Δt_{init}	1 s (15 minutes)	Initial time increment, Δt [T]. The recommended value for the initial time step depends on the type of simulation and boundary conditions used. When simulating a process that starts with a large initial pressure head or concentration gradient at the boundary (e.g., ponded infiltration or a sudden change of boundary concentration), use a small value of the initial time step (e.g., 1 s). When simulating a long-term process with variable boundary conditions (e.g., seasonal or multiyear simulation), start with a larger time step (e.g., 15 min). This is because this initial time step is used whenever time variable boundary conditions significantly change (e.g., the water flux changes by 25% or more). If needed (if there is no convergence for Δt_{init}), the program will still use a smaller time step than Δt_{init}, but starting with a larger Δt_{init} leads to more efficient calculations. In general, smaller initial time steps must be used for soil with more nonlinear soil hydraulic properties (e.g., course textured soils), and larger initial time steps can be used for soil with less nonlinear soil hydraulic properties (e.g., loam).
Δt_{min}	1 s	Minimum permitted value of the time increment, $\Delta t_{\min}$ [T]. The minimum time step must be smaller than a) the initial time step, b) the interval between print times, and c) the interval between time-variable boundary condition records. Always specify a small minimum allowed time step, on the order of 1 s. This value may never be used, but it provides the code with flexibility when needed, e.g., when there is a sudden change in boundary fluxes, and HYDRUS may not converge with larger time steps.
Δt_{max}	Large	Maximum permitted value of the time increment, Δt_{max} [T]. This is a relatively unimportant parameter, and a large value may be specified. Since HYDRUS automatically selects its optimal time step, there is usually no need to constrain that. The only time when there is a need to constrain the time step is likely when HYDRUS is asked to generate internally intra-daily variations in temperature or evaporation and transpiration fluxes. Then, there is a need to have time steps smaller (e.g., 1 hour) to model these daily variations properly.
It_{crit}	10	Maximum number of iterations allowed during any time step while solving the nonlinear Richards equation using a modified Picard method. The recommended and default value is 10 (or even as low as 8). It is usually not helpful to use a larger value than 10. If HYDRUS does not converge in 10 iterations, then there is a relatively small probability that it will do so during more iterations. Even if it does, it is much more efficient to reduce the time step and attempt to find the solution with a smaller time step, which is done automatically by the program when It_{crit} is reached.
It_{min}	3	Optimal value in most cases.
It_{max}	7	Optimal value in most cases.
k_1	1.3	Optimal value in most cases. Only when there is a saturated zone in the profile, e.g., a perched water layer, the numerical solution may be more stable with smaller k_1 (e.g., 1.1).
k_2	0.7	Optimal value in most cases.

Additionally, the time step is also constrained by the stability requirements of the numerical solution of the convection-dispersion solute transport equation:

a) The local Courant numbers are calculated for each finite element and used to limit the time step so that the local Courant numbers are always lower than one:

$$Cr^e = \frac{q\Delta t}{\theta R \Delta x} \le 1$$

$$\Delta t \le \frac{\theta R \Delta x}{q}$$

b) The performance index ω_s is evaluated, and the time step is adjusted so that the performance index remains below a specified limit:

$$Pe \bullet Cr \le \omega_s \quad \text{(e.g., = 2)}$$

$$\frac{q\Delta x}{\theta D}\frac{q\Delta t}{\theta R \Delta x} = \frac{q^2 \Delta t}{\theta^2 DR} \le \omega_s$$

$$\Delta t \le \omega_s \frac{\theta^2 DR}{q^2}$$

In HYDRUS, it is possible to vary some variables versus time internally. For example, HYDRUS can generate daily or annual changes in temperature using a sine wave with a prescribed mean temperature and a temperature amplitude. Similarly, HYDRUS can also generate variations of transpiration during the day (see the technical manual). In such cases, when time changes of selected variables are generated internally by HYDRUS, it is up to users to limit the maximum time step so that the effects of these changes can be adequately modeled. For these internally generated time-variable changes (e.g., in temperatures and/or ET), limiting the maximum time step to about 1/20 of one time period (e.g., the time step of 0.05 d for a daily cycle) is recommended.

2. Spatial Discretization

2.1. FE-Mesh in HYDRUS-1D

The finite element mesh is constructed by dividing the soil profile into linear elements whose sizes are defined by the *z*-coordinates of the nodes that form the element corners.

2.2. FE-Mesh in HYDRUS (2D/3D)

The finite element mesh is constructed by dividing the flow region for two-dimensional problems into quadrilateral and/or triangular elements or for three-dimensional problems into tetrahedral, hexahedral, and/or triangular prismatic elements whose shapes are defined by the coordinates of the nodes that form the element corners. The numerical program automatically subdivides the quadrilaterals into triangles (or hexahedrals and triangular prisms into tetrahedrals), which are then treated as subelements.

2.3. Common Rules

Generally, the finer the FE-mesh size, the more precise the results will be. On the other hand, the number of calculation steps increases (for more FE nodes) as additional equations must be solved for every further FE node. This affects the computing time considerably. A mesh size that is too fine slows down the calculation without significantly improving the quality of the analysis. On the other hand, a coarse mesh may not satisfyingly capture the boundary conditions, and the numerical solution may not converge.

Finite element dimensions must be adjusted to a particular problem. They should be made relatively small at locations where large hydraulic gradients are expected. Such a region is usually located close to the soil surface, where highly variable meteorological factors can cause rapid changes in the soil water content and corresponding pressure heads. Similarly, regions with sharp gradients can be located near the internal sources or sinks. Hence, we recommend using relatively small elements at and near the soil surface. The size of elements can gradually increase with depth to reflect the generally much slower changes in pressure heads at deeper depths. We also recommend using elements of approximately equal size to decrease numerical errors. The ratio of the sizes of two neighboring elements is not recommended to exceed about 1.5.

The required size of finite elements close to the soil surface depends greatly on the specified boundary conditions. When boundary conditions are specified for daily or shorter time intervals, usually resulting in short-duration fluxes of a large magnitude, spatial discretization needs to be finer (on the order of cm) than when boundary conditions are specified for longer time intervals (e.g., weekly or monthly).

The element dimensions should also depend upon the soil hydraulic properties. For example, coarse-textured soils having relatively high *n*- and α-values generally require a finer discretization than fine-textured soils. That is because their soil hydraulic functions are more nonlinear, and thus, the numerical solution may be less stable. To demonstrate this issue, we have carried out simulations of ponded infiltrations into sand and clay soil profiles (see Figure 1). Notice that the

pressure head front for a sandy profile is very sharp, and the entire front is only about 5 cm thick. To describe this front using our numerical model, we need several FE nodes at the front, implying that our spatial discretization must be on the order of 1 cm or less. On the other hand, the pressure head (and correspondingly water content) front for a clay profile is relatively smooth, and consequently, our spatial discretization can be much coarser.

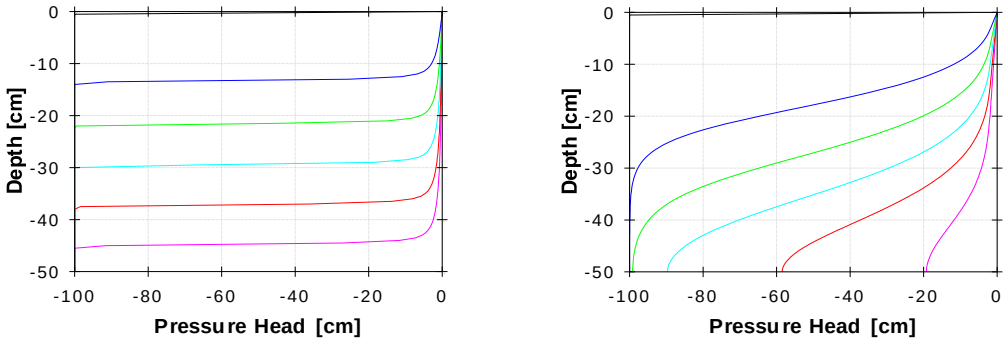
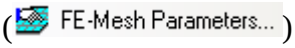


Figure 1. Pressure head profiles for ponded infiltration into sand and clay soil profiles.

2.4. HYDRUS (2D/3D) Rules

In higher dimensions, it is often recommended that FE elements have approximately all sides equal to obtain smooth solutions. This recommendation is not valid for most applications involving fluxes in the vadose zone. Since vertical fluxes usually dominate over horizontal fluxes in the vadose zone, the spatial discretization must be much finer in the vertical direction than in the horizontal direction. In general, the spatial discretization should reflect the expected gradients in the transport domain; it should be fine in the direction of large gradients and can be coarser in the direction of small gradients.

HYDRUS offers several parameters and tools to adjust the spatial discretization to expected flow and transport conditions. The four most important parameters and tools are the **Targeted FE Size**, the **Smoothing Factor**, **Mesh Stretching**, and the **FE-Mesh Refinement**. The first three parameters can be specified in the **FE Mesh Parameters** dialog window, which can be displayed by clicking on the menu command "Edit->FE-Mesh->FE-Mesh Parameters" or the **FE-Mesh Parameters** command (

) at the Tools Sidebar when the **FE-Mesh View** is displayed.

HYDRUS uses two different FE-Mesh generation modules. It uses the **MeshGen** module to generate FE meshes for two-dimensional domains and the **Base Surface** of three-dimensional layered domains. It then uses the **Genex/T3D** module to generate FE meshes for three-dimensional general domains (note that three-dimensional general domains are available only in the 3D-Professional Level of HYDRUS). These are relatively different approaches to generating FE meshes; thus, using the above-discussed parameters and tools is sometimes also different. In the text below, we will attempt to emphasize these differences. One can select a preferred generator on the **Mesh tab** of the **Program Options** dialog.

2.4.1. Targeted FE Size

The **Targeted FE Size** (see Figure 2) defines the global FE mesh size, i.e., the size of finite elements that the FE-Mesh generator will try to reach throughout the transport domain. FE-Mesh can be made finer or coarser in different parts of the transport domain by defining **FE-Mesh Refinements** (see discussion below) for these parts of the domain.

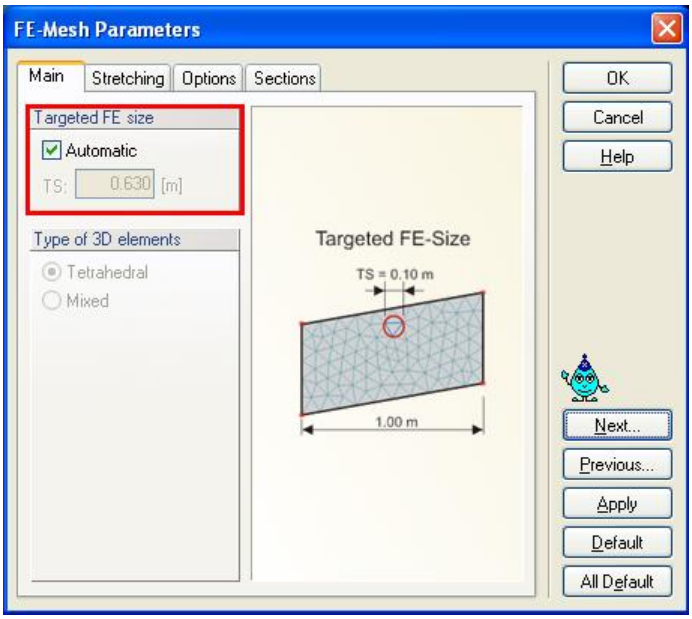


Figure 2. The **FE-Mesh Parameters** dialog window with the **Main Tab** displayed.

Note that the **Targeted FE Size** suggested by HYDRUS (when the check box **Automatic** is on) is related only to the size of the transport domain and is a certain fraction (e.g., 1/60) of the diameter of the transport domain. The purpose of this number is to suggest the size of finite elements so that the number of elements is reasonable. The **Automatic Targeted FE Size** is not adjusted to any other factors involved in a particular project, such as the nonlinearity of soil hydraulic properties, how the boundary conditions are defined, heterogeneity of the soil profile, and so on. Users need to adjust the **Targeted FE Size** for their particular problems themselves and then adjust the FE-mesh where needed using **FE-Mesh Refinements** as needed.

Note that the **Targeted FE Size** is used by **GENEX** in all three dimensions and by **MeshGen** in two dimensions (for the **Base Surface** of Layered Three-Dimensional domains). Distances between layers in Layered Three-Dimensional domains are defined differently (see below).

2.4.2. Smoothing Factor

The **Smoothing Factor** (see Figure 3) is the ratio of maximum and minimum heights of a finite element triangle. The **Smoothing Factor** is used only for FE-meshes generated using **MeshGen**, i.e., 2D and 3D-Layered domains, but not for FE-meshes generated using **Genex**, i.e., for 3D-General domains. For a triangle with equal sizes, this factor is equal to 1 (which is theoretically not achievable for finite element meshes). The **Smoothing Factor** can be decreased to a value of about 1.1 when a highly smooth finite element mesh is required and, vice-versa, can be increased when a course mesh can be tolerated. The **Smoothing Factor** significantly affects the final number of elements, with the number of elements decreasing dramatically for larger values of the smoothing factor (e.g., 3). We recommend keeping the **Smoothing Factor** at its default value (i.e., 1.3) for relatively small transport domains (e.g., simulations of irrigation details) and increasing its value to about 2-3 for larger transport domains (e.g., soil transects, field scale simulations) (see Figure 4).

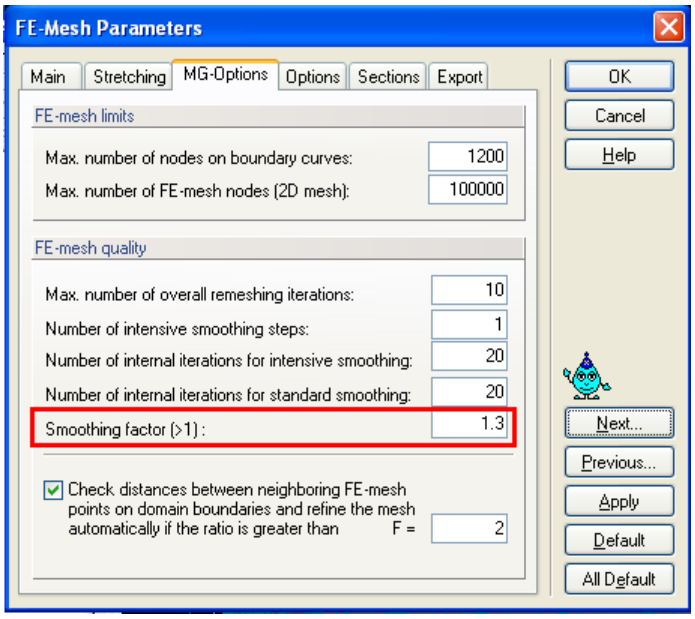


Figure 3. The **FE-Mesh Parameters** dialog window with the **MeshGen-Options Tab** displayed.

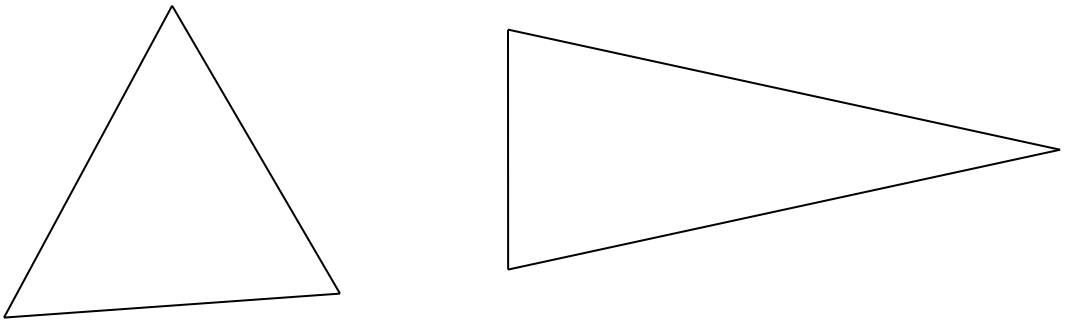


Figure 4. Triangles with a smoothing factor equal to 1 (left) and 3 (right).

2.4.3. Stretching

Stretching of the finite element mesh (i.e., the degree of mesh anisotropy in a certain direction) is defined using the **Stretching Factor** and the **Stretching Direction** (see Figure 5). The finite elements are made larger in the particular **Stretching Direction** if the **Stretching Factor** is larger than one and smaller if it is less than one. This transformation results in a mesh deformed in the given direction, which can be desirable for problems that, for example, require different spatial steps (mesh sizes) in the *X*, *Y*, and *Z* directions. As discussed above, in the vadose zone, vertical fluxes usually dominate over horizontal fluxes. Thus, the spatial discretization must be much finer in the vertical direction than in the horizontal direction. If vertical fluxes are expected to be many times larger (e.g., 10-100 times) in the vertical direction than in the horizontal direction, the **Stretching Factor** can also be very large (e.g., 10-100) (see Figure 6).

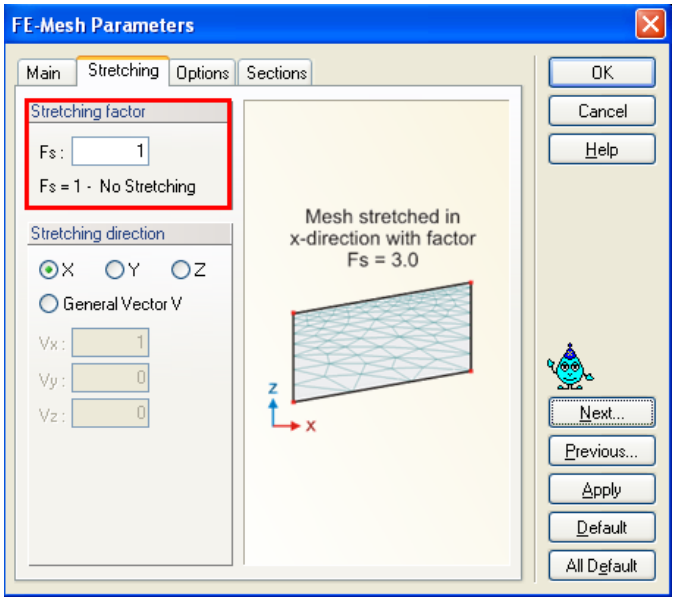


Figure 5. The **FE-Mesh Parameters** dialog window with the **Stretching Tab** displayed.

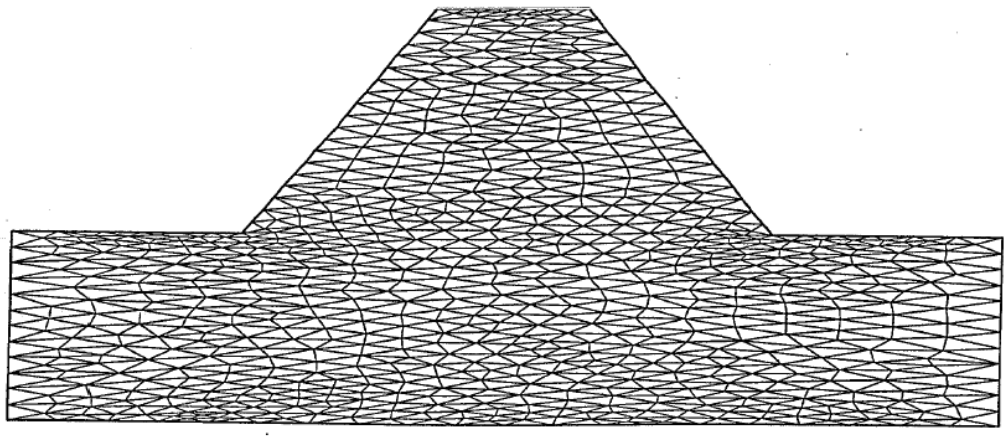


Figure 6. Example of mesh stretching using a stretching factor of 3 in the *x*-direction.

2.4.4. FE-Mesh Refinement

If no **FE-Mesh Refinements** have been defined, the FE mesh is generated with the preset **Targeted FE size**. The global FE mesh settings are described in Chapter 2.4.1.

The mesh configuration can be affected by **FE-Mesh Refinements** that are set for selected parts of the computational domain. Use this option to refine the mesh where needed, such as in parts of the computational domains where large pressure head and/or concentration gradients are expected. Large pressure head and/or concentration gradients usually occur close to the soil surface (due to variable atmospheric fluxes), close to internal sinks/sources (drains, drippers, wells), or near interfaces between different soil layers with significantly different soil hydraulic properties.

Using **FE-Mesh Refinements**, one may try to find an appropriate compromise between the accuracy of the results and the computational time. Basically, there are four types of **FE-Mesh Refinements**:

- Refinement around a node
- Refinement on a line
- Refinement on a surface
- Refinement on a solid (used only with Genex)

a) FE-Mesh Refinements in the MeshGen Module

Figure 7 shows the **New FE-Mesh Refinement** dialog windows used with the **MeshGen** module for four different types of refinements. **FE-Mesh Refinements** can be applied to a **Point**, a **Line**, and a **Surface**.

When the **FE-Mesh Refinement** is assigned to a **Point** (Figure 7 top left), users only need to define the finite element size to be used at a given point. This FE size will be used around a given point, while sizes of neighboring FE will be gradually increased until the **Targeted FE Size** is reached further away in the computational domain. Notice that the dialog window also displays the **Global Targeted FE Size**. The same **FE-Mesh Refinement**, i.e., the same refined FE size, can be assigned to multiple **Points** in the transport domain.

There are two ways in which the **FE-Mesh Refinement** can be assigned to a **Line** (Figure 7, top right and bottom left). Users can specify the **FE-Size** (**Line – FE-Size;** Figure 7 top right) or the **Number of Points** (**Line – Number of Points**; Figure 7 bottom left) for the entire **Line**. Similarly, as for points, the **FE-Size** (either given directly or calculated from the number of points) will be used on the **Line**, while sizes of FE away from the **Line** will gradually increase until the **Targeted FE Size** is reached further away in the computational domain. Again, the same **FE-Mesh Line Refinement**, i.e., the same refined FE size, can be assigned to multiple **Lines** in the computational domain.

The **FE-Mesh Refinement** can also be assigned to a selected **Surface** (Figure 7 bottom right). In this case, users need to define FE-Size for a selected **Surface**. Note that this option is useful only when multiple **Surfaces** are used in the computational domain. The refined FE-Size is then used on the entire surface, including boundaries with other **Surfaces**, and the **Targeted FE Size** is reached only further away in the other **Surfaces** of the computational domain.

The **Point** or **Surface** with an assigned **FE-Mesh Refinement** is indicated using the following marks:  and  , respectively. **FE-Mesh Refinement** is marked on curves using green points. Contrary to other green points on the curves, they can be selected.

Figure 7. The **FE-Mesh Refinement** dialog windows for the **MeshGen** module with four different types of refinements (applied to a **Point**, a **Line** with a given FE-size or the number of Points, and to a **Surface**).

b) FE-Mesh Refinements in the Genex Module

While the **MeshGen** module discretizes the computational domain into unstructured FE-mesh using triangles (in 2D) and tetrahedrals (in 3D), the **Genex** module uses primarily quadrilateral (in 2D) and hexahedral (in 3D) finite elements. **Genex** uses triangles only in parts of the domain where needed, e.g., when it refines the grid. Although quadrilateral or hexahedral elements may be generated by **Genex**, the computational module subdivides these elements into triangles or tetrahedrals, respectively. While the transition between domains with refined FE-mesh and domains with global mesh size in **MeshGen** is relatively smooth and gradual (see Figure 7), in **Genex,** the extent of this transition zone is defined exactly by users (see Figures below). Similarly to **MeshGen**, the same **FE-Mesh Line Refinements** can be assigned to multiple **Points**, **Lines**, **Surfaces**, and **Solids** in the computational domain.

There are two ways in which the **FE-Mesh Refinement** can be assigned to a **Point** (Figure 8). Users can use either **Circular** or **Rectangular** refinement. For a **Circular Refinement** around a point, a radial refinement area is defined around a node in all directions. Users need to specify the **Radius** of the refinement area, the **Inner Targeted FE-Size** (i.e., an FE size immediately around a point), and the **Outer Targeted FE-Size** (i.e., an FE size at the outer end of the radial refinement area). The **Outer Targeted FE-Size** should be equal to or slightly smaller than the **Global Targeted FE-Size**. When generating the FE mesh, the mesh refinement is carried out gradually towards the center because the FE length at the periphery of the refinement area is usually identical to the global mesh width. For larger differences between the inner and outer FE length, it is recommended to set the radius sufficiently large (not too small) to avoid acute-angled triangular finite elements in the refinement area. A rectangular refinement area is defined around a node in all directions for a **Rectangular Refinement**. Users need to specify the **Side Length** of the refinement area and the **Inner Targeted FE-Size** (i.e., an FE size immediately around a point).

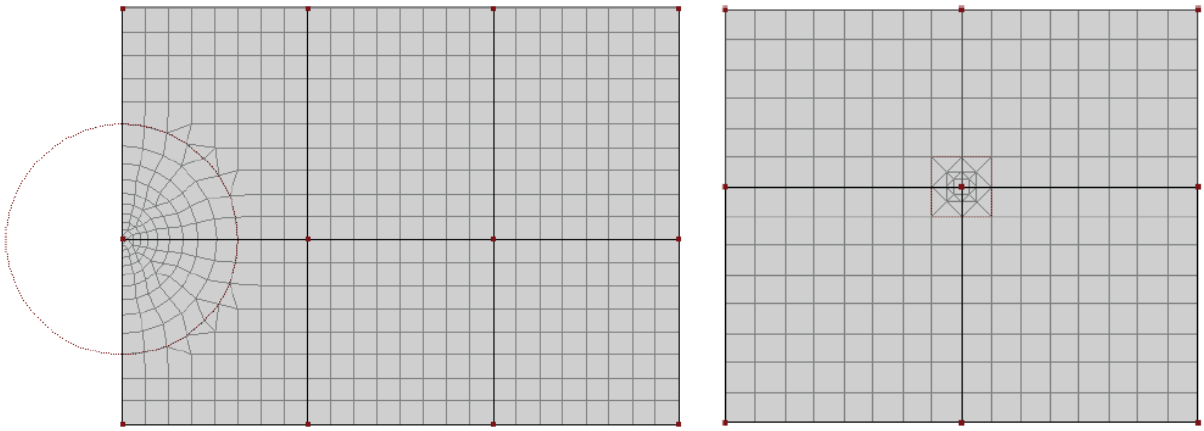


Figure 8. Circular (left) and rectangular (right) refinements around a node.

FE-Mesh Refinements assigned to a **Line** are handled in **Genex** similarly to **MeshGen**. One can again define the size (spacing; **Line through FE-Length**) or the number (a specific number of equidistant sub-divisions; **Line through Division**) of finite elements along a line. In the

former case, one needs to define the **Target FE Length**, and in the latter case, the **Number of Division FE Nodes**. However, in **MeshGen,** there was a gradual increase in the sizes of finite elements away from the **Line**. In **Genex,** the refinement affects only one row of finite elements (Figure 9).

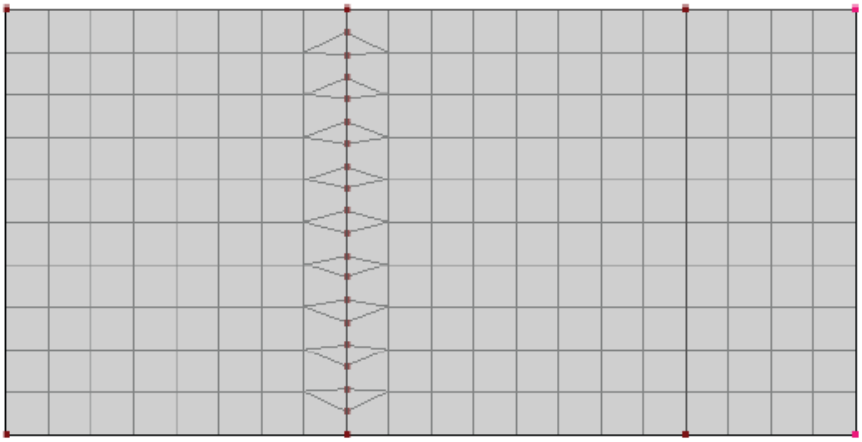


Figure 9. Refinement on a line (by defining the size or the number of finite elements along a line).

It is also possible to define **Fe-Mesh Refinements** for **Surfaces** or **Solids** (Figure 10). In both cases, users need to specify the **Targeted FE-Size**, which will be set as the target size of the finite elements for the entire surface or solid.

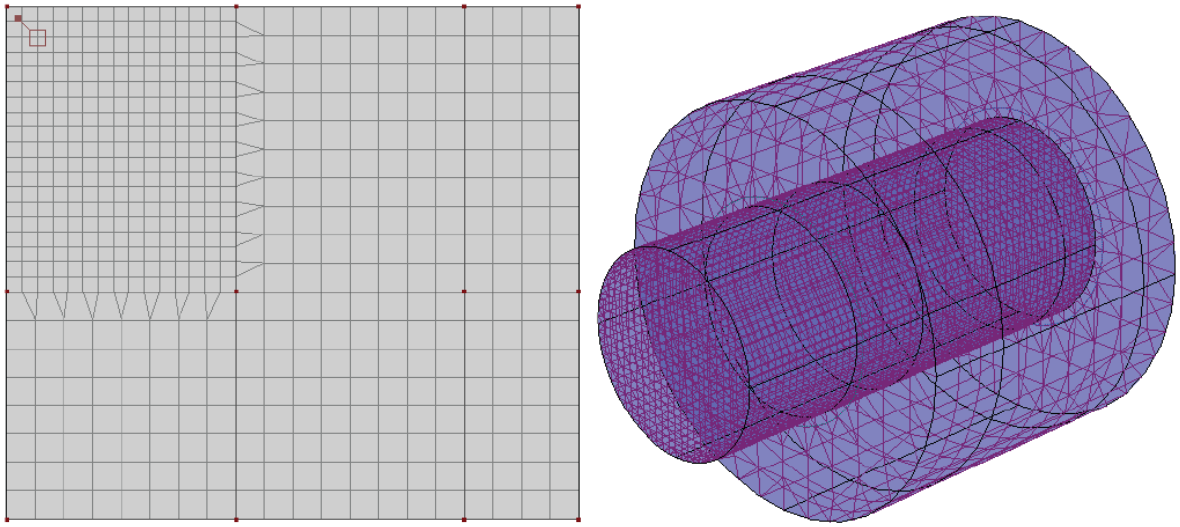
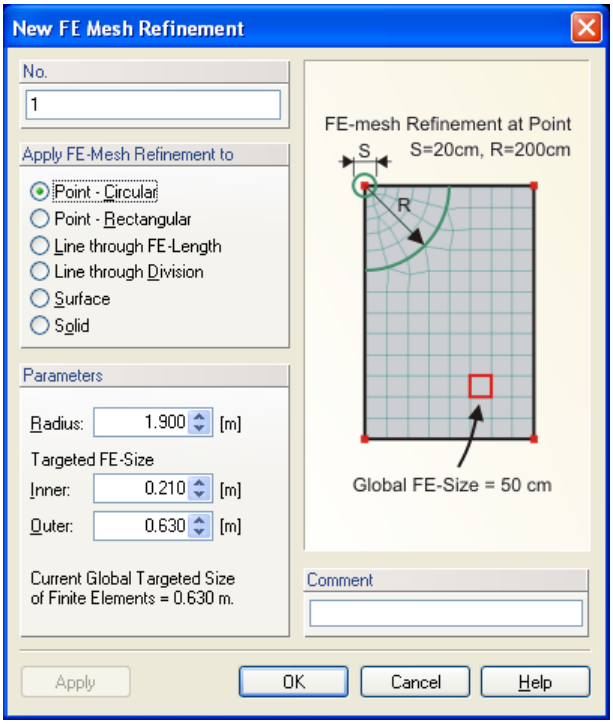
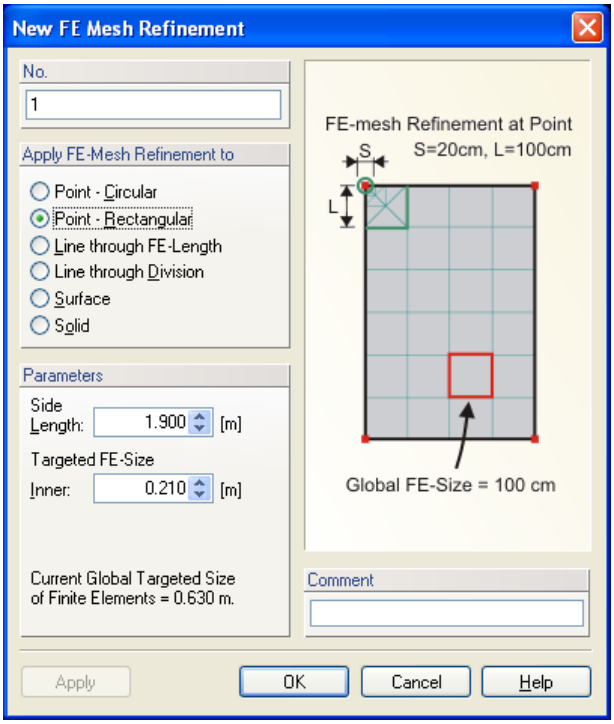
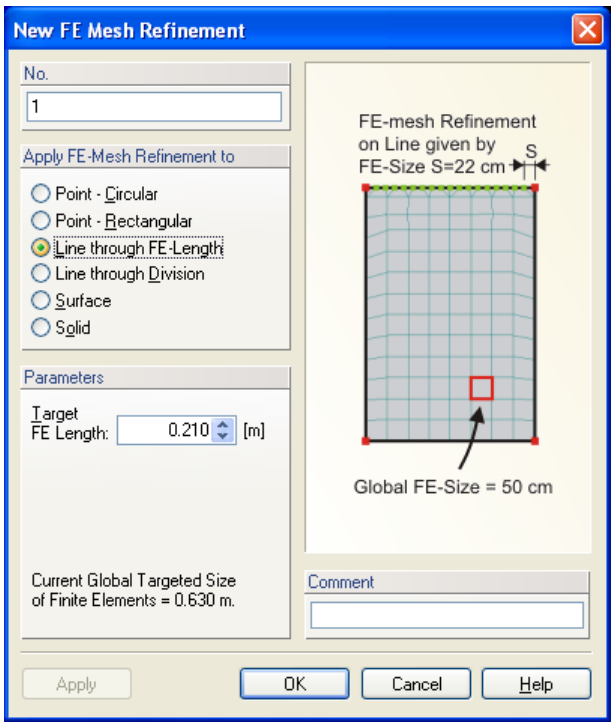
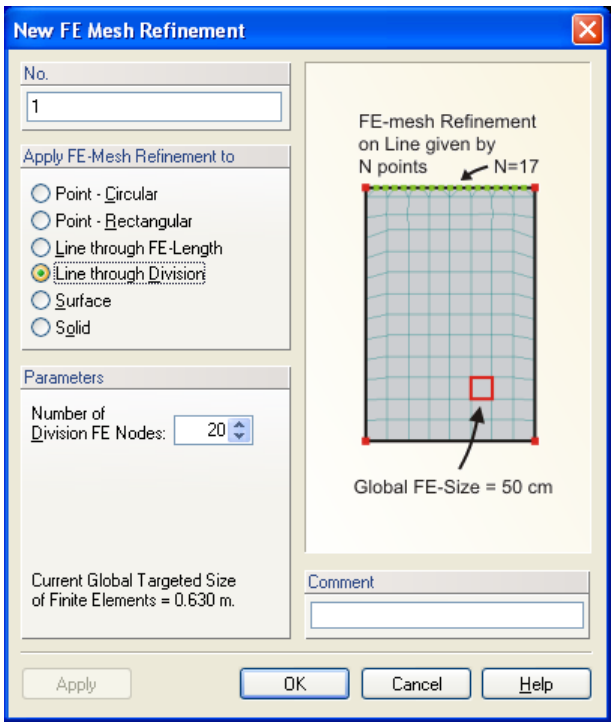


Figure 10. Refinements on a surface (left) or solid (right).


New FE Mesh Refinement
No.
1
Apply FE-Mesh Refinement to
Point - Circular
Point - Rectangular
Line through FE-Length
Line through Division
Surface
Solid
Parameters
Radius: 1.900 [m]
Targeted FE-Size
Inner: 0.210 [m]
Outer: 0.630 [m]
Current Global Targeted Size of Finite Elements = 0.630 m.
FE-mesh Refinement at Point
S=20cm, R=200cm
S
R
Global FE-Size = 50 cm
Comment
Apply
OK
Cancel
Help


New FE Mesh Refinement
No.
1
Apply FE-Mesh Refinement to
Point - Circular
Point - Rectangular
Line through FE-Length
Line through Division
Surface
Solid
Parameters
Side Length: 1.900 [m]
Targeted FE-Size
Inner: 0.210 [m]
Current Global Targeted Size of Finite Elements = 0.630 m.
FE-mesh Refinement at Point
S=20cm, L=100cm
S
L
Global FE-Size = 100 cm
Comment
Apply
OK
Cancel
Help


New FE Mesh Refinement
No.
1
Apply FE-Mesh Refinement to
Point - Circular
Point - Rectangular
Line through FE-Length
Line through Division
Surface
Solid
Parameters
Target FE Length: 0.210 [m]
Current Global Targeted Size of Finite Elements = 0.630 m.
FE-mesh Refinement on Line given by FE-Size S=22 cm
S
Global FE-Size = 50 cm
Comment
Apply
OK
Cancel
Help


New FE Mesh Refinement
No.
1
Apply FE-Mesh Refinement to
Point - Circular
Point - Rectangular
Line through FE-Length
Line through Division
Surface
Solid
Parameters
Number of Division FE Nodes: 20
Current Global Targeted Size of Finite Elements = 0.630 m.
FE-mesh Refinement on Line given by N points
N=17
Global FE-Size = 50 cm
Comment
Apply
OK
Cancel
Help

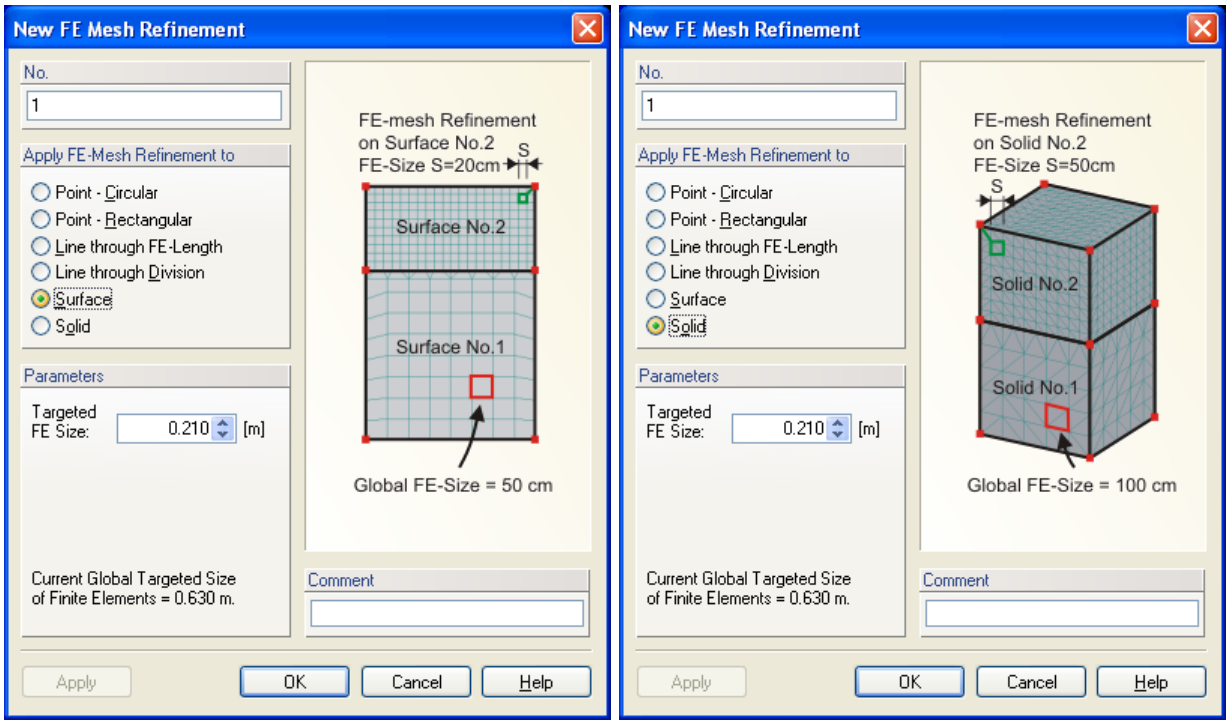


Figure 11. The **FE-Mesh Refinement** dialog window for the **Genex** module with six different types of refinements (applied to a **Circular** or a **Rectangular Point**, a **Line** with a given FE-size or a number of points, to a **Surface,** and to a **Solid**).

2.4.5. Vertical Discretization of 3D Layered Domains

3D-Layered Domains (**Solids**) are formed by a **Base Surface** (discretized into finite elements using the **MeshGen** module) and one or multiple **Thickness Vectors**. A **Solid**, i.e., its **base surface** and **thickness vectors**, is defined and can be edited in the **Edit Solid** dialog window with four tabs: **General** (Figure 12), **Sub-Layers**, **Thickness Profiles**, and **FE-Mesh**. The **General Tab** provides information on which **Base Surface** and **Thickness Vectors** define the solid. Vertical discretization when the **Base Surface** is in the horizontal plane or horizontal discretization when the **Base Surface** is in the vertical plane is then defined in the other three tabs (i.e., **Sub-Layers**, **Thickness Profiles**, and **FE-Mesh**).

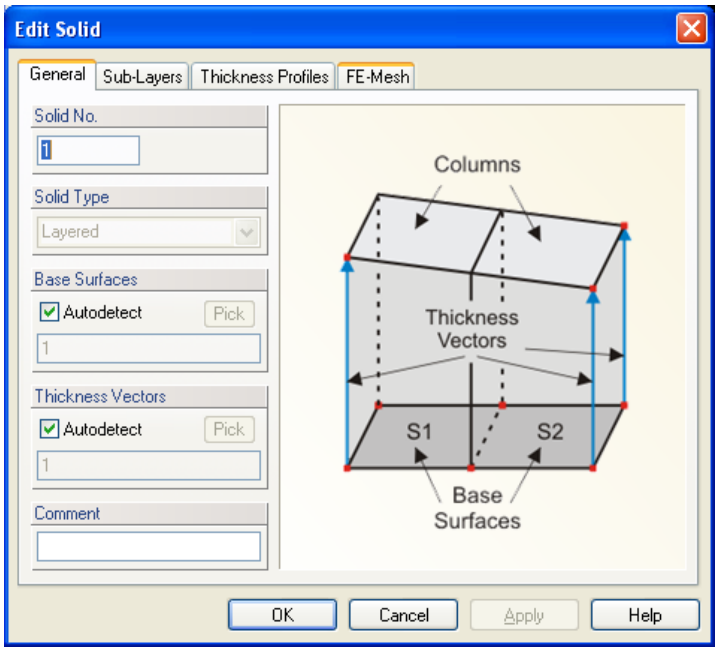


Figure 12. The **Edit Solid** dialog window with the **General Tab** displayed.

The **Sub-Layers Tab** (Figure 13) informs whether the solid is divided into one or more **Sub-Layers**. **Layers** are other objects that can be used to subdivide a single solid. For example, these layers can be used to keep constant thicknesses of selected horizons or constant discretization close to the soil surface (to get good estimates of evaporation) across the entire transport domain (solid). **Sub-Layers** are defined on a single **Thickness Vector**, the so-called **Master Thickness Vector**.

In the **Edit Solid** dialog, it is possible to define the number of **Sub-Layers** and their **Thicknesses**. A **Solid** always has one **Master Thickness Vector**, one of the **Thickness Vectors** of a **Solid** that has a special meaning, as described below.

The **Thickness** of a **Sub-Layer** is calculated as follows:

- **Thicknesses of Sub-Layers** given in the Table are calculated on the **Master Thickness Vector**

- The sum of the **Thicknesses of all Sub-Layers** should equal the length of the **Master Thickness Vector**. If not, a program will automatically issue a warning and recalculate the **Thicknesses of Sub-Layers**.
- A **Solid** can have more **Thickness Vectors** of different lengths so that specified **Thicknesses of Sub-Layers** can be maintained. The program then does the following: For **Sub-Layers** with the **Constant Thickness Type**, the specified thickness is maintained at all **Thickness Vectors**, i.e., over the entire computational domain. For **Sub-Layers** with the **Variable Thickness Type**, their thicknesses are linearly increased or decreased so that the sum of the **Thicknesses of all Sub-Layers** corresponds with the length of a particular **Thickness Vector**.

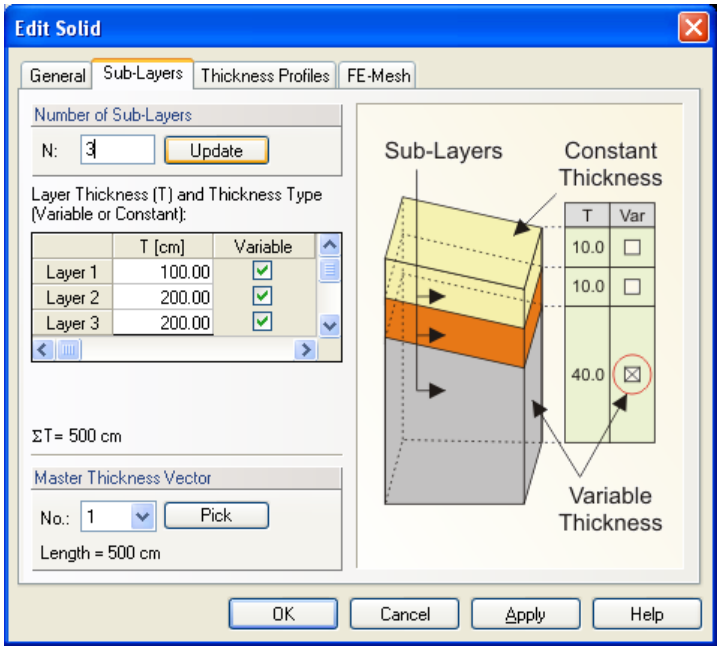


Figure 13. The **Edit Solid** dialog window with the **Sub-Layers** Tab displayed.

In the preceding paragraph, we described how to define the **Thicknesses of Sub-Layers** on the **Master Thickness Vector** using a table. This table represents the so-called **Master (Default) Profile**, i.e., a particular distribution of thicknesses. Suppose one wants to define precisely the division of thicknesses on vectors other than the **Master Thickness Vector**. It is then necessary to create additional **Thickness Profiles** and use them on corresponding **Thickness Vectors**. One **Thickness Profile** (**Default Profile**) is created automatically by the code. Figure 14 shows a dialog (**the Thickness Profiles Tab** of the **Edit Solid** dialog) to create additional **Thickness Profiles**. There is always a **Default Profile**, corresponding to the table described in Figure 13 above. One can create new profiles, change their thicknesses, or delete them. One can simultaneously also see a list of **Thickness Vectors** where the selected **Thickness Profile** is used.

These additional **Thickness Profiles** must be subdivided into the same number of **Sub-Layers** as the **Default Profile,** and these **Sub-Layers** can have either constant or variable thickness (**T**) across the transport domain. Thicknesses and the mode (constant or variable) of particular layers

are specified in a table. The thickness Sum (**TT**) is then calculated by adding the thicknesses of particular layers. At least one layer thickness must be variable. The finite element discretization then follows these layers.

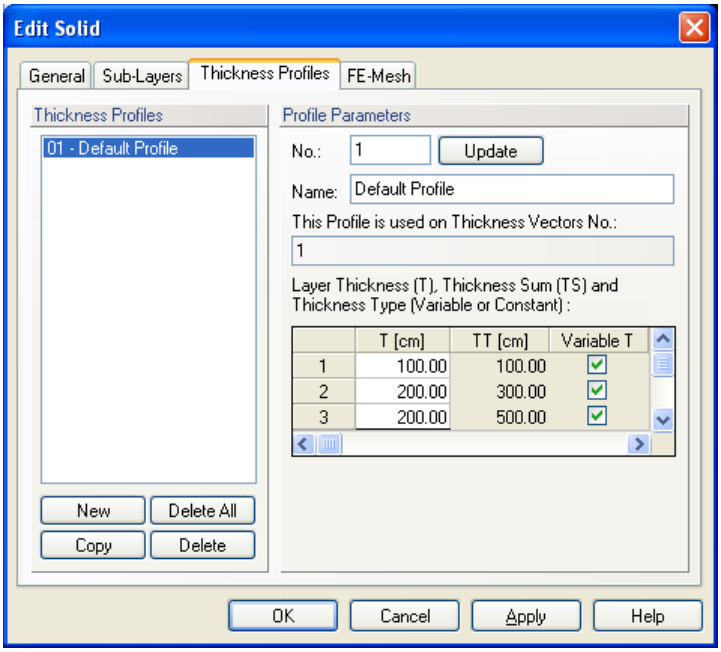


Figure 14. The **Edit Solid** dialog window with the **Thickness Profiles Tab** displayed.

Finally, the **FE-Mesh Tab** (Fig. 15) specifies how many horizontal FE-Layers are used to discretize the solid. When only one **Sub-Layers** (Figure 13) exists, then users can specify relative finite element spacing (spacing of vertical discretization layers) on the vertical side (***FE-Mesh Layer Spacing***) using the RS1 (relative size at the top) and RS2 (relative size at the bottom) factors below ***Generate Mesh Layer Spacing***. The element sizes are then proportionally distributed. The preview part of the dialog window shows the main terms used on each Tab. When multiple **Sub-Layers** (Figure 13) exist, users can specify the relative sizes of elements for each layer (***FE-Mesh Density in Layers***).

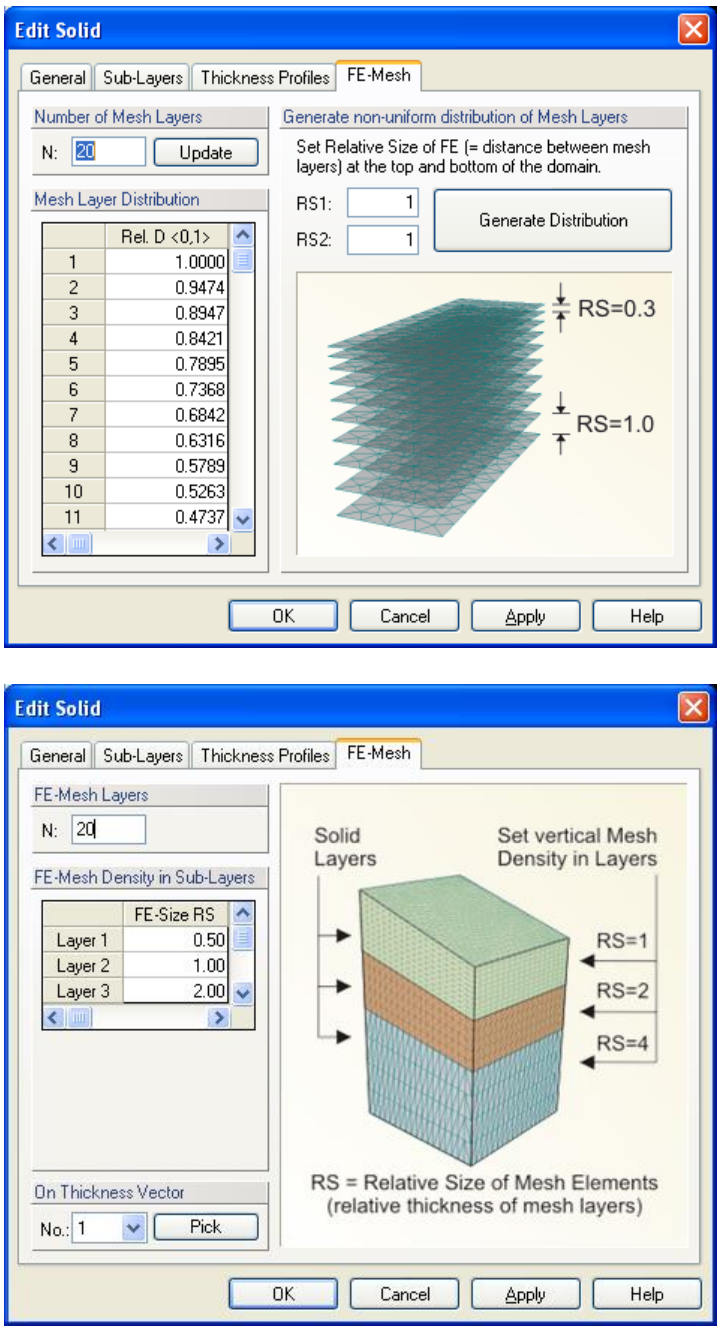


Figure 15. The **Edit Solid** dialog window with the **FE-Mesh Tab** displayed for a single (top) and multiple layers (bottom).

2.4.6. FE-Mesh Generation

Select the **Generate FE-Mesh** command (Generate FE-Mesh) in the **Calculation** menu or at the Tools Sidebar to start the FE mesh generation.

It is recommended that the generated FE-mesh be checked before the calculation is started. In this way, you can determine whether efficiently refined mesh has been created or if refinement areas are still required. It is also possible that a coarser FE-mesh covers areas of minor interest when evaluating the result. You can define, for example, a **FE-Mesh Refinement** with a **Targeted FE-Size** larger than the defined target size. This speeds up both analysis and results output.

If the FE mesh generation has succeeded, select the **FE-Mesh Statistic** command (FE-Mesh Statistics...) in the **Calculation** menu or at the Tools Sidebar to open a dialog box with information about the generated FE mesh.

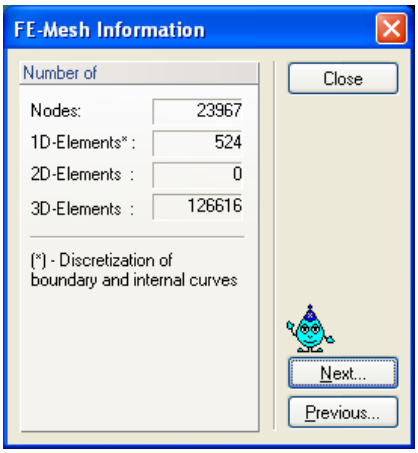


Figure 16. The **FE-Mesh Information** dialog window.

The statistics show the types and numbers of the generated finite elements, which helps select the appropriate matrix solver method and judging the approximate calculation time.

To delete the mesh, select the **Delete FE-Mesh** command (Delete FE-Mesh) in the **Calculation** menu or at the Tools Sidebar. When you use this function, all results that may be available will be deleted, too.