# Modelling of three-dimensional, transient groundwater flow

# GGU-3D-TRANSIENT

**VERSION 4**

# Contents

# 1 Preface

The **GGU-3D-TRANSIENT** program allows modelling of transient groundwater flow in three-dimensional groundwater systems using the finite element method. The program includes a powerful mesh generator and routines (contours, 3D graphics, etc.) for comfortable evaluation of the modelling results.

Data input is in accordance with conventional WINDOWS operations and can therefore be learned almost entirely without the use of a manual. Graphics output supports the true-type fonts supplied with WINDOWS, so that excellent layout is guaranteed. Colour output and graphics (e.g. files in formats BMP, JPG, PSP, TIF, etc.) are supported. PDF and DXF files can also be imported by means of the integrated Mini-CAD module (see the "**Mini-CAD**" manual).

It is not the aim of this manual to offer an introduction to the finite-element method. For details of the finite-element method, please see O. C. Zienkiewicz, "*Methode der Finiten Elemente*" ("Finite-Element Methods"), Carl Hanser Verlag Munich Vienna, 1984. The hydraulic basics used in the FEM program can be taken from, e.g., J. Buß "*Unterströmung von Deichen*" (Announcement 92, Leichtweiß-Institut, TU Braunschweig).

The program system has been used in a large number of projects and has been thoroughly tested (using analytical solutions and in comparison with other FEM applications). No faults have been found. Nevertheless, liability for completeness and correctness of the program and the manual, and for any damage resulting from incompleteness or incorrectness, cannot be accepted.

The following general notes are important:

- The data basis is represented by steady-state systems imported from the **GGU-3D-SSFLOW** program.

- Triangular elements are used.

- Darcy's Law applies.

- The piezometric heads are calculated in a linear manner for each element.

## 2   Licence protection and installation

In order to guarantee a high degree of quality, a hardware-based copy protection system is used for the **GGU-3D-TRANSIENT** program.

The GGU software protected by the *CodeMeter* copy protection system is only available in conjunction with the *CodeMeter stick* copy protection component (hardware for connection to the PC, "*CM stick*"). Because of the way the system is configured, the protected software can only be operated with the corresponding CM stick. This creates a fixed link between the software licence and the CM stick copy protection hardware; the licence as such is thus represented by the CM stick. The correct Runtime Kit for the CodeMeter stick must be installed on your PC.

Upon start-up and during running, the **GGU-3D-TRANSIENT** program checks that a CM stick is connected. If it has been removed, the program can no longer be executed.

For installation of GGU software and the CodeMeter software please refer to the information in the *Installation notes for GGU Software International*, which are supplied with the program.

## 3   Language selection

**GGU-3D-TRANSIENT** is a bilingual program. The program always starts with the language setting applicable when it was last ended.

The language preferences can be changed at any time in the "**Info**" menu, using the menu item "**Spracheinstellung**" (for German) or "**Language preferences**" (for English).

# 4    Program concept

An initial steady-state condition is required for time-dependent modelling of groundwater flow. **GGU-3D-TRANSIENT** acquires this initial condition from a record (e.g. ATEST.3fd_is) generated during a previous modelling using the **GGU-SS FLOW3D** application. It is therefore necessary to load this file first after opening the program. Beside the results, this file also contains the soil properties upon which the steady-state modelling is based. These *steady-state* soil properties ($k$, $n_{eff}$, etc.) cannot be altered from within the **GGU-3D-TRANSIENT** program. If amendment of these values were allowed, the soil properties would no longer correlate to the calculated potentials and discharges and the transient modelling would produce nonsensical results. However, additional soil properties, which are required for transient modelling, must be provided.

Further to these additional soil properties, time-dependent boundary conditions can also be defined with **GGU-3D-TRANSIENT**. You must define boundary values (potential or discharge) as a function of time, by means of polygon courses. These polygon courses can then be assigned system nodes, either individually or in groups. Soil properties (transient) and polygon courses, as well as their allocations to system nodes, can be saved in a separate file (known as boundary data), in order to make them available again for later modelling.

Boundary conditions defined in the course of steady-state modelling (**GGU-3D-SSFLOW**) are also contained in the file (e.g. ATEST.3fd_is). They are assumed by the program to be constant for the whole modelling duration, unless you delete the so-called *fixed* boundary conditions in **GGU-3D-TRANSIENT**.

The **GGU-3D-TRANSIENT** program is started using conventional WINDOWS operations. The program is equipped with a large amount of *error queries*. Nonsensical input will usually be caught by the program and an error message displayed on the screen. Independently of this you should, for security reasons, intermittently save longer input, if only to save renewed input in the case of a power cut.

# 5 Starting the program

After starting the program, you will see two menus at the top of the window:

- File

- Info

After clicking on the "**File**" menu, a system processed with the **GGU-3D-SSFLOW** program can be loaded by means of the "**Load**" menu item.

> The processed system record is saved in **GGU-SS FLOW3D** using the "**.f3d_is**" extension, in order to make it available for **GGU-3D-TRANSIENT**. The program function to be activated for this purpose can be found in **GGU-3D-SSFLOW** in the "**File/Export to GGU-3D-TRANSIENT**" menu item.

After selecting the system, the menu bar will show eleven menus:

- File

- Mesh(x/y)

- Time BC

- Fixed BC

- System

- Graphics preferences

- Page size + margins

- Times

- Evaluation

- 3D

- Info

After clicking one of these menus, the so-called menu items roll down, allowing access to all program functions.

The program works on the principle of *What you see is what you get*. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the **GGU-3D-TRANSIENT** screen is not refreshed after every alteration.

If you would like to refresh the screen contents, press either [**F2**] or [**Esc**]. The [**Esc**] key additionally sets the screen presentation back to your current zoom, which has the default value 1.0, corresponding to an A3 format sheet.

# 6 Short description

Since I know from personal experience that reading user-manuals is quite a chore, there will now follow a short description of the main program functions. After studying this section you will be in a position to model transient groundwater flow. You can learn about the details of the program in the later chapters.

- In order to be able to use the system entered in your **GGU-3D-SSFLOW** groundwater flow program for transient modelling using the **GGU-3D-TRANSIENT** program, you must create a file in the **GGU-3D-SSFLOW** program with the file extension "**. f3d_is**" an. You can create this file by going to "**File/Export to GGU-3D-TRANSIENT**".

- Start the **GGU-3D-TRANSIENT** program and, using the menu item "**File/Load**", select the system from which you would like to model transient groundwater flow.

- Then, using the "**File/Load boundary values**" menu item, either load boundary values defined in a previous sitting into the system or specify new boundary conditions using the menu "**Time BC**" or "**Fixed BC**".

- Using the "**Time BC/Soil properties**" menu item, you can query the soil properties which you defined in **GGU-3D-SSFLOW**. This list is supplemented by information on the specific storage coefficient. According to soil type, you can define your specific soil curves, using modelling points by clicking on the "**nw = f(u)**" field (*cf*. Figure 1 further below). The soil permeability can be altered as a function of the time using the "**k = f(t) [0]**" button.

- Using the menu item "**Time BC/Edit polygon coursees**" you specify the temporal change in, e.g., a potential boundary condition. This is initially independent of location. By clicking on the menu item you will see a dialog box within which, by clicking on the "**New**" button and then "**Edit polygon course**", you can specify the number of polygon points, i.e. the number of temporal changes in your potentials.

- Assign system nodes a boundary condition polygon using "**Time BC/Point potential or source**".

- With the help of the "**System/Model**" menu item the program model the transient groundwater flow with the user-defined boundary conditions. You can specify the final time and the temporal increment in the dialog box.

- For visualisation and evaluation of the modelling results proceed to either the "**Times**", "**Evaluation**" or "**3D**" menu. You can almost always leave the subsequent dialog boxes using the "**OK**" button, without having to make further changes. The program will usually suggest sensible input. The "**Determine extreme values**" switch, which appears in a number of dialog boxes, should be clicked once however, otherwise an error message will appear. You can also use the tools in the *3D toolbar* directly to evaluate and visualise your modelling results.

- If you have a colour printer correctly installed in WINDOWS, you can create colour output by going to "**File/Print and export**" and then pressing the "**Printer**" button in the subsequent dialog box. For black and white printers, grey scales will be used.

This short description demonstrates that only a few items need be selected to modell a transient groundwater system. All further menu items deal principally with saving data, layout and further evaluation of the model, where required. The description will ensue in the following chapters.

# 7 Description of menu items

## 7.1 File menu

### 7.1.1 "Load" menu item

In **GGU-3D-TRANSIENT**, you can import the data from a previous modelling of a steady-state, three-dimensional groundwater model created using **GGU-3D-SSFLOW**. This file has the standard extension "**. f3d_is**" and is created in **GGU-3D-SSFLOW** using the menu item "**File/Export to GGU-3D-TRANSIENT**".

### 7.1.2 "Save" menu item

You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved <u>without prompting</u> with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

### 7.1.3 "Save as" menu item

You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use "**. f3d_is**" as file suffix, as this is the suffix used in the file requester box for the menu item "**File/Load**". If you choose not to enter an extension when saving, "**. f3d_is**" will be used automatically.

### 7.1.4 "Load boundary values" menu item

Using this menu item you can reload a "**.dat3d**" file containing boundary values which was previously defined and saved with the "**File/Save boundary values**" menu item, in order to further process this or to use it as the basis for transient analysis. The file contains the *transient* soil properties and the polygon courses, as well as the allocations of the polygon courses to the system nodes.

### 7.1.5 "Save boundary values" menu item

After selecting this function you can save the defined boundary values in order to have the data available again at a later date. The file contains the *transient* soil properties and the polygon courses, as well as the allocations of the polygon courses to the system nodes.

### 7.1.6 "Export to ASCII file" menu item

If an FEM mesh has been generated, the node coordinates can be saved to an ASCII file,. The x and y coordinates of the underlying mesh (2D) are saved and, in addition, if the system has been modelled, the potential of the active plane.

### 7.1.7 "Printer preferences" menu item

You can edit printer preferences (e.g. swap between portrait and landscape) or change the printer in accordance with WINDOWS conventions.

### 7.1.8 "Print and export" menu item

You can select your output format in a dialog box. You have the following possibilities:

- **"Printer"**
  allows graphic output of the current screen contents. to the WINDOWS standard printer or to any other printer selected using the menu item "**File/Printer preferences**". But you may also select a different printer in the following dialog box by pressing the "**Printer prefs./change printer**" button.
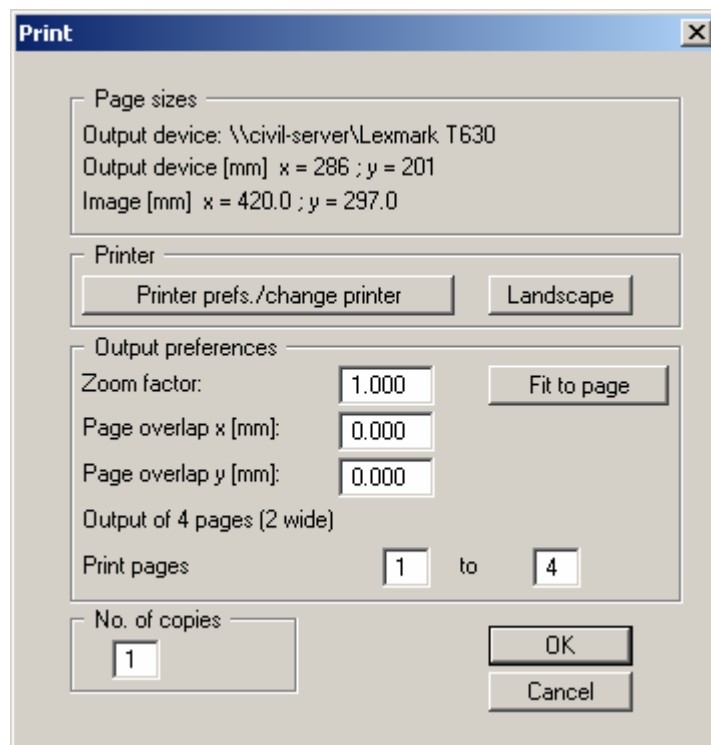
The upper group box of the dialog box shows the maximum dimensions which the printer can accept. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate better re-connection of the images, the possibility of entering an overlap for each page, in x and y direction, is given. Alternatively, you also have the possibility of selecting a smaller zoom factor, ensuring output to one page ("**Fit to page**" button). Following this, you can enlarge to the original format on a copying machine, to ensure true scaling. Furthermore, you may enter the number of copies to be printed.

- **"DXF file"**
  allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.

- **"GGUCAD file"**
  allows output of the graphics to a file, in order to enable further processing with the **GGUCAD** program. Compared to output as a DXF file this has the advantage that no loss of colour quality occurs during export.

---

- **"Clipboard"**
  The graphics are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further processing, e.g. into a word processor. In order to import into any other WINDOWS program you must generally use the "*Edit/Paste*" function of the respective application.

- **"Metafile"**
  allows output of the graphics to a file in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

---

If you select the **"Copy/print area"** tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see **"Tips and tricks"**, Section 8.3).

Using the **"Mini-CAD"** program module you can also import EMF files generated using other GGU applications into your graphics.

---

- **"MiniCAD"**
  allows export of the graphics to a file in order to enable importing to different GGU applications with the **Mini-CAD** module.

- **"GGUMiniCAD"**
  allows export of the graphics to a file in order to enable processing in the **GGUMiniCAD** program.

- **"Cancel"**
  Printing is cancelled.

### 7.1.9 "Batch print" menu item

If you would like to print several appendices at once, select this menu item. You will see the following dialog box:
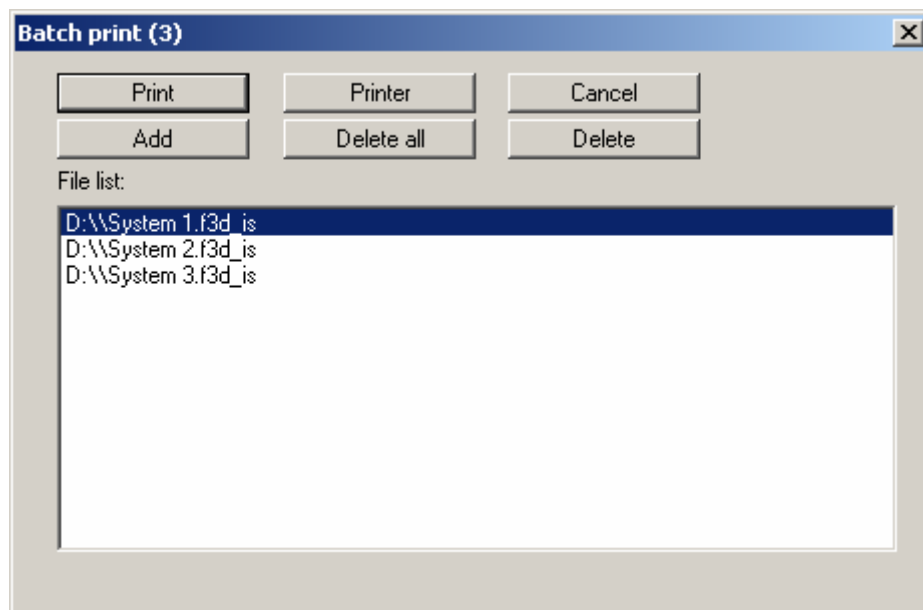


Create a list of files for printing using "**Add**" and selecting the desired files. The number of files is displayed in the dialog box header. Using "**Delete**" you can mark and delete selected individual files from the list. After selecting the "**Delete all**" button, you can compile a new list. Selection of the desired printer and printer preferences is achieved by pressing the "**Printer**" button.

You then start printing by using the "**Print**" button. In the dialog box which then appears you can select further preferences for printer output such as, e.g., the number of copies. These preferences will be applied to all files in the list.

### 7.1.10 "Exit" menu item

After a confirmation prompt, you can quit the program.

### 7.1.11 "1, 2, 3, 4" menu items

The "**1, 2, 3, 4**" menu items show the last four files worked on. By selecting one of these menu items the listed file will be loaded. If you have saved files in any other folder than the program folder, you can save yourself the occasionally onerous *rummaging* through various sub-folders.

#### 7.2.1    "Preferences" menu item

In the dialog box for this menu item you specify how the FEM mesh for your system or the outline of the FEM mesh is displayed on the screen. The element no. and the soil no. cannot be displayed simultaneously. The preferences are accepted after closing the box using "**OK**".

#### 7.2.2    "Mesh" menu item

If you go to this menu item or click the ⬡ tool in the toolbar, the FEM mesh is displayed.

If you had previously defined a section to evaluate your transient model, use this menu item to return to the selected plane of the system visualisation. To select a different plane click the arrow buttons in the *3D toolbar* (see Section 7.6.8).

#### 7.2.3    "Outline" menu item

If you go to this menu item or click the ⬠ tool in the toolbar, your system outline is displayed. To select a different plane click the arrow buttons in the *3D toolbar* (see Section 7.6.8).

### 7.3.1    General notes on time-dependent boundary conditions

Using this menu item you can define the *time-dependent* boundary conditions for the system and specify soil properties if this has not already been carried out in **GGU-3D-SSFLOW**.

### 7.3.2    "Preferences" menu item

This menu item allows the font sizes for node and polygon course labelling to be edited. In addition, you may specify whether the user-defined polygon courses are to be labelled and define the formatting.

### 7.3.3    "Soil properties" menu item

After going to this menu item a system-specific dialog box opens allowing input of the *transient* soil properties. The *steady-state* properties imported from the **GGU-3D-SSFLOW** program can no longer be amended here.



The steady-state soil properties are supplemented by the specific storage coefficient of the fluid and the grain skeleton "**Ss**". The soil permeability can be altered as a function of the time using the "**k = f(t) [0]**" button.

The value for $n_w$ can also be entered here. The value of $n_w$ gives the volume of water relative to the total volume. It is a dimensionless variable and is a function of the pore water pressure u. Typical function curves for various soil types are shown in Figure 1:
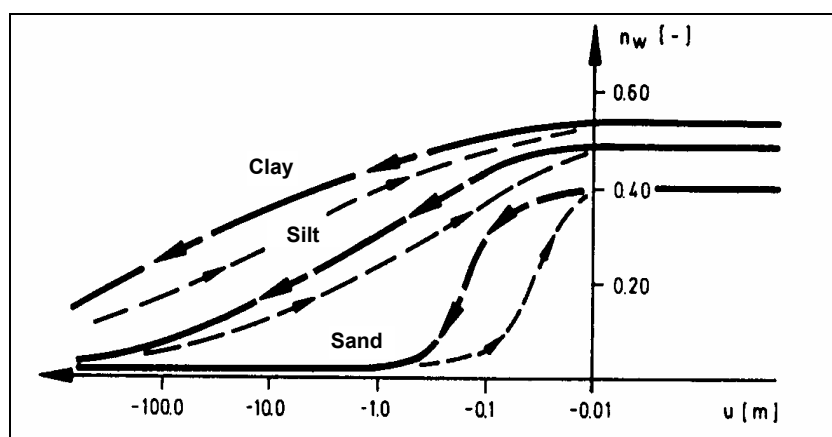


*Figure 1  Function $n_w = f(u)$*

By clicking the "**nw = f(u)**" button you will arrive at the following dialog box:



The table offers you the possibility of developing the specific soil curve of the soil to be processed by assigning various pore water pressures u to $n_w$ values. By clicking on the "**x modelling points to edit**" function box you can edit the number of modelling points. The entered values are adopted for all soil layers after an additional confirmation if the "**For all**" button is clicked.

Alternatively to manual input, the default values for the various soil types (gravel, sand, silt, clay) can be automatically adopted by the program by pressing the "**Std**" button in the above "**Soil properties**" dialog box.

### 7.3.4 "Edit polygon courses" menu item

The temporally variable (transient) system boundary conditions for the system are defined using this menu item, or previously defined polygon courses may be edited. Input is initially location-independent. After clicking an existing polygon course or the "**New**" button, a dialog box opens providing the following options:

- **"Done"**
  Once all changes to the selected polygon course are complete, exit the dialog box using this button. By clicking the "**New**" button once again further polygon courses can be defined.

- **"Edit polygon course"**



You define the number of polygon course modelling points via the "**2 polygon points to edit**" button. It is possible to define up to 100 polygon points. Once you have done this you can define the temporal course with your corresponding boundary condition value. Take care to note the units you are using. If you have entered the permeability in m/s, you must also enter the time values in seconds. Boundary conditions for sources must then be entered in m³/s and potentials in m. The example uses seconds. It has proven useful for evaluation purposes to use m/day for permeability as this provides much more manageable times. In principle, however, there are no restrictions imposed, as the program always functions true to your dimensions. If you would like to paste a polygon point, enter this as an additional value at the end of the table and then select the "**Sort**" button.

If you enter polygon values smaller than $-10^{20}$, no potential boundary conditions will be applied for subsequent potential node allocations intended for transient modelling. You can thus suppress a potential boundary condition for given times.

> When defining polygon courses, avoid sudden leaps which do not correspond to natural conditions. Otherwise a ***pressure shock*** may be the result, leading to the necessity for extremely small time increments to arrive at a solution.

After clicking on the "**Done**" button, you will return to the initial dialog box.

- **"View polygon course"**
  Using this button you have presented your polygon course in a diagram for viewing and checking.

- **"Delete polygon course"**
  By clicking this button you can delete the current polygon course after a confirmation prompt.

- **"Modify polygon course"**
  You can modify the polygon course values globally using addition, subtraction, multiplication and division.

- **"Load polygon course"**
  The data from a previously saved file can be loaded (e.g. from a previous project or a different system).

- **"Save polygon course"**
  The data entered can be saved to a file in order to have them available for different systems, for example.

- **"Import ASCII data"**
  If you have polygon course data available as an ASCII file, they can be imported into the program.

- **"Export ASCII data"**
  The current polygon course can be exported to an ASCII file.

- **"Duplicate polygon course"**
  The data for the current polygon course are duplicated and allocated to a new polygon course. You are then moved directly to the dialog box for the new polygon course and are able to view, edit or delete the data by means of the corresponding buttons there (e.g. "**x polygon points to edit**", see above).

- **"Polyg. course via sine funct."**
  This allows you to generate a polygon course via a sine function. This can be useful for areas under tidal influence for example.

- **"Copy polygon course to rear"**
  Existing polygon course points can be copied and appended to the end. This shortens input, for example if two *waves* follow each other.

### 7.3.5 "Delete all" menu item

You can delete all polygon courses.

### 7.3.6 "Point potential or source" menu item

Once one or more polygon courses have been defined, they can be assigned individual FEM nodes. To do this, simply click the node to which a polygon is to be assigned.
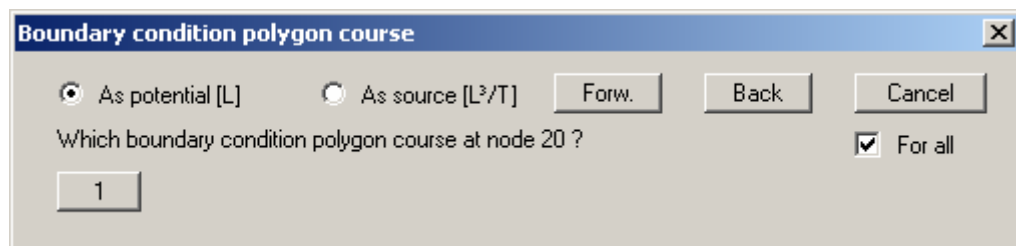


Define the type of boundary condition to use by activating either of the option buttons "**As potential**" or "**As source**". If the "**For all**" check box is activated the boundary condition at this point is adopted for all planes. Then specify the polygon course number for allocation to the marked FEM node by clicking the corresponding button. In the system visualisation, this FEM node is labelled with the polygon course number and additionally with a "**P**" or "**Q**" to indicate a potential or a source.

A defined potential or source can be deleted by clicking it again using the right mouse button.

### 7.3.7 "(Potential or source) In section" menu item

After going to this menu item, time-dependent potentials or sources can be defined or deleted for several FEM nodes simultaneously by assigning them a polygon course. Define the section by clicking four points in an anti-clockwise direction. Use the right mouse button to undo. The definition refers to all nodes within the quadrilateral. If the boundary condition is to be applied to all planes, activate the "**For all**" check box. If the "**Define**" button is pressed the same dialog box opens as for "**Time BC/Point potential or source**" (see Section 7.3.6).

### 7.3.8 "(Potential or source) Linear" menu item

After going to this menu item click the nodes of the FEM mesh to which time-dependent, linearly variable potentials or sources are to be assigned by means of a polygon course. Press the [**Return**] key and enter a start and an end polygon course. The program subsequently generates new polygon courses by linear interpolation of the spacing of the system nodes and assigns these to the corresponding nodes. This menu item can be useful for defining a receiving watercourse if its water level is linearly variable within the systems.

### 7.3.9 "Diffuse sources" menu item

Certain elements can be assigned time-dependent diffuse sources by defining the area required as a polygon using 4 anti-clockwise points. You will then see the following dialog box:
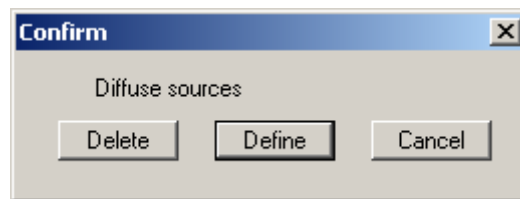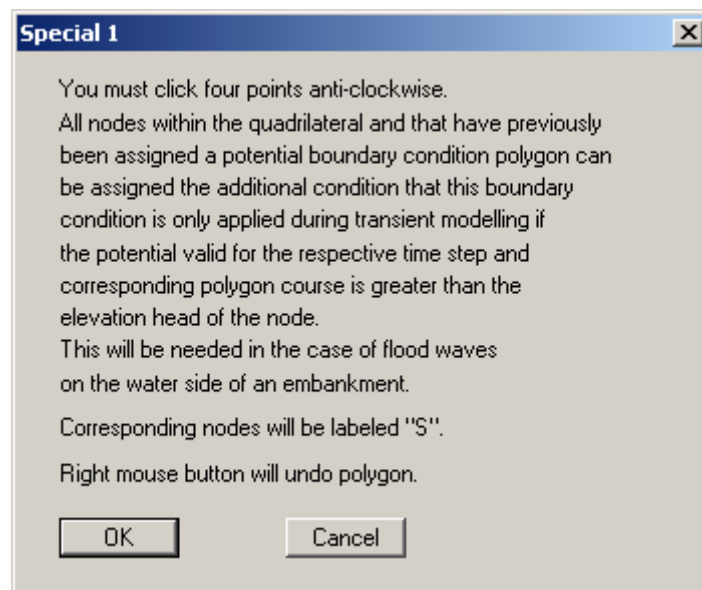


If diffuse sources have already been defined they can be deleted from the marked elements by pressing the "**Delete**" button. After clicking the "**Assign**" button a dialog box opens for assigning a polygon course. This menu item is required for simulation of a rainfall event, for example. During transient modelling the program first determines the time-dependent value for the polygon course and multiplies this with the element area. This is subsequently assigned as a source to the three nodes of an element. In the system visualisation, the FEM element is labelled with the polygon course number and additionally with an "**f**".

### 7.3.10 "Display all polygon courses" menu item

You can have all user-generated polygon courses displayed as hydrographs in a single image. Using the dialog box you can specify the type of presentation.

### 7.3.11 "Special 1" menu item

A message box provides information on this function after clicking on the menu item.
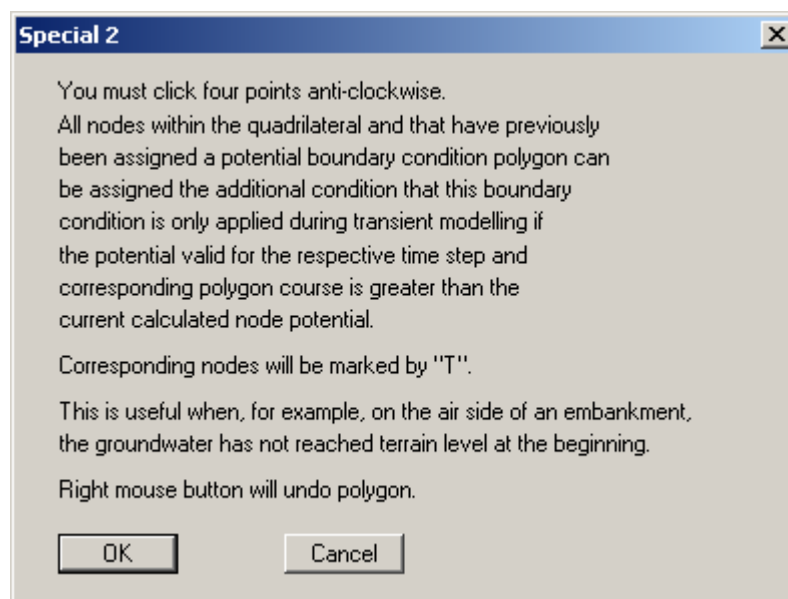
**Special 1**

You must click four points anti-clockwise.
All nodes within the quadrilateral and that have previously
been assigned a potential boundary condition polygon can
be assigned the additional condition that this boundary
condition is only applied during transient modelling if
the potential valid for the respective time step and
corresponding polygon course is greater than the
elevation head of the node.
This will be needed in the case of flood waves
on the water side of an embankment.

Corresponding nodes will be labeled "S".

Right mouse button will undo polygon.

[OK]        [Cancel]

You can thus restrict the occurrence of boundary conditions This can be useful in order to avoid a *negative pressure* in the system.

For example, this can apply to the water side of an embankment if a potential is initially defined at the top of the embankment and then the water level lowered to the base.

### 7.3.12 "Special 2" menu item

The message box explains the functions.

**Special 2**

You must click four points anti-clockwise.
All nodes within the quadrilateral and that have previously
been assigned a potential boundary condition polygon can
be assigned the additional condition that this boundary
condition is only applied during transient modelling if
the potential valid for the respective time step and
corresponding polygon course is greater than the
current calculated node potential.

Corresponding nodes will be marked by "T".

This is useful when, for example, on the air side of an embankment,
the groundwater has not reached terrain level at the beginning.

Right mouse button will undo polygon.

[OK]        [Cancel]

### 7.3.13    "Special 3" menu item

A further message box informs you of the functions in this menu item.
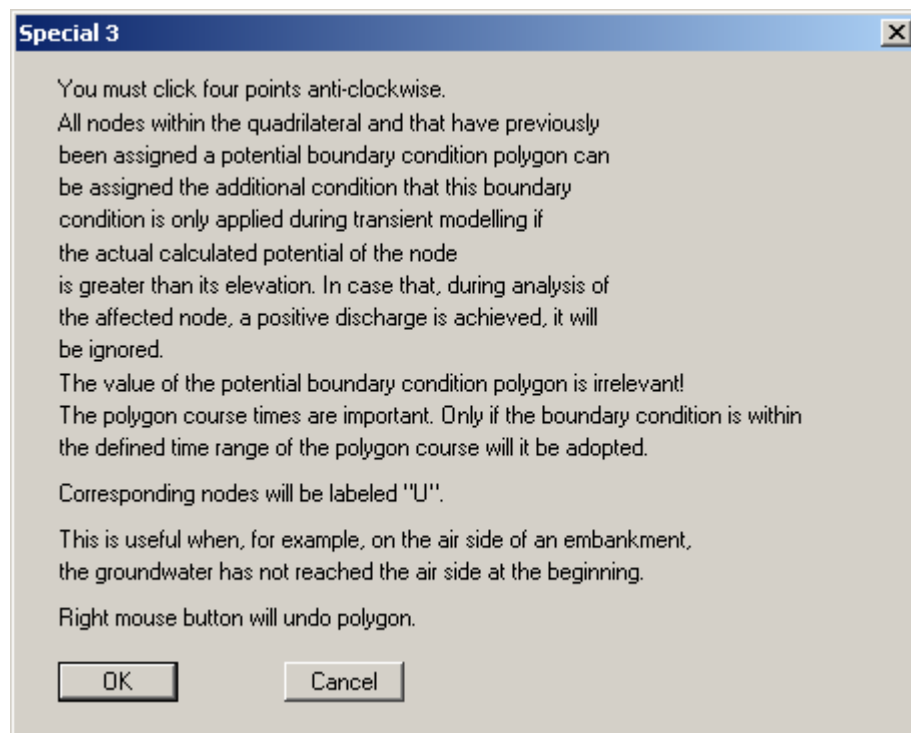


## 7.4    Fixed BC menu

### 7.4.1    General notes on "fixed" boundary conditions

The **GGU-3D-TRANSIENT** program also imports the steady-state boundary conditions from the steady-state modelling record (henceforth also known as *fixed* boundary conditions). When analysing transient systems it is almost always the case that at least parts of these boundary conditions remain constant for the complete investigation duration. In principle, you could define the appropriate boundary condition polygons and assign nodes to these. However, this is onerous and superfluous. For this reason the program initially assumes all steady-state boundary conditions as being temporally constant. Naturally, if nodes are then assigned a boundary condition polygon, the temporal constancy at this node is deleted. If a node boundary condition is not to be applied during transient modelling, you must delete the *fixed* boundary condition (see below).

> Do not misuse the *fixed* boundary conditions to assign a substantially different boundary condition value than that which applied in the steady-state condition. Otherwise a *pressure shock* may be the result, leading to the necessity for extremely small time increments to arrive at a solution. It is better to define a boundary condition polygon for the appropriate nodes with smooth transitions, which correspond better to the naturally occurring conditions.

### 7.4.2 "Preferences" menu item

Here you can specify preferences for the type of presentation for *fixed* boundary potentials or fixed sources at the node points. This includes the font size selection and the type of numerical presentation (decimal places).

### 7.4.3 "Point potentials" menu item

After selecting this menu item a descriptive dialog box opens, followed by the screen presentation of the system mesh with the previously defined *fixed* boundary conditions. By clicking with the **right** mouse button you can delete fixed boundary conditions, similarly to the time boundary conditions. By clicking with the **left** mouse button you can define potentials, similarly to the time boundary conditions, which remain constant for the modelling duration.
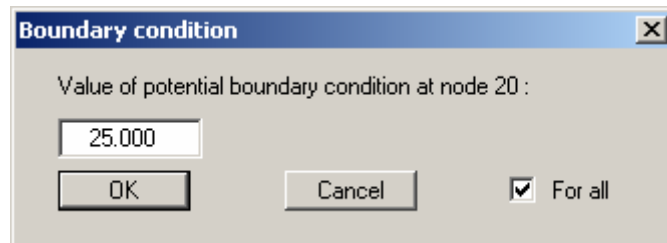
In the dialog box you define the value and type of boundary condition. If the "**For all**" check box is activated the boundary condition at this point is adopted for all planes. However, please observe the remark above (see Section 7.4.1).

### 7.4.4 "(Potentials) In section" menu item

After selecting this menu item several FEM nodes can be simultaneously assigned a potential boundary condition in a defined section, or an existing boundary condition be deleted. For this click four points in an anti-clockwise direction using the left mouse button. The right mouse button will undo. The definition refers to all nodes within the quadrilateral.
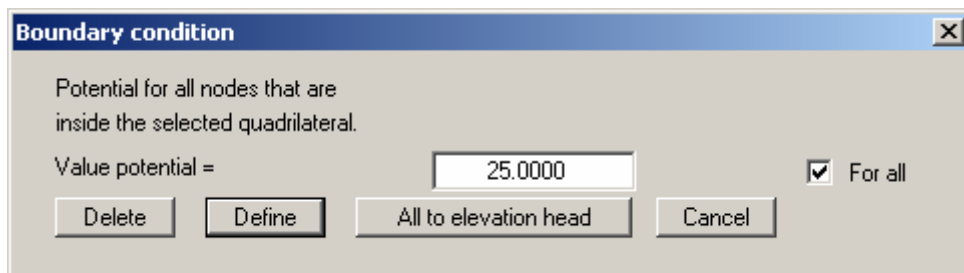
You can now define the potentials or delete all marked potentials. The definition applies to the nodes located within the section. In addition, by pressing "**All to elevation head**", it is possible to define the potentials in the enclosing polygon as being at the elevation head entered for the system. If the "**For all**" check box is activated the boundary condition at the marked points are adopted for all planes.

### 7.4.5 "(Potentials) Linear" menu item

After going to this menu item you click the nodes of the FEM mesh to which a potential boundary condition is to be assigned. Press [**Return**] to then enter a start and an end potential and, if necessary, activate the "**For all**" check box in order to apply the boundary conditions at the marked points for all planes.

The selected node is assigned a potential which increases linearly from the start potential to the end potential, proportional to the separation of the nodes to each other. This menu item is helpful if you need to consider a water course, for example, the water level of which changes linearly within the system.

### 7.4.6 "Point sources" menu item

Using this menu item source boundary conditions are defined within the FEM system in analogy to the fixed potential boundary conditions (see Section 7.4.3). A source is positive if it brings water into the system.

### 7.4.7 "Linear sources" menu item

After going to this menu item you define a polygon by clicking 4 points anti-clockwise. The element nodes having their centroid within the thus marked quadrilateral are assigned a linear source with the user-defined value, d, d. If an element lies entirely within the quadrilateral, the corresponding nodes are not assigned a linear source.

If you need to delete existing linear sources, define the polygon again and enter **0**.

### 7.4.8 "Diffuse sources" menu item

Certain elements can be assigned diffuse sources by defining the area required as a polygon using 4 counter-clockwise points. You will then see the following dialog box:
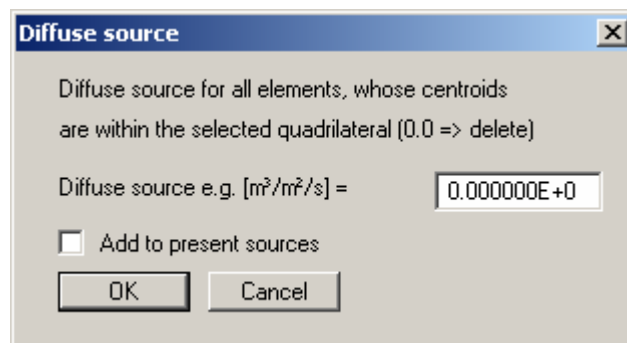


If you activate the "**Add to present sources**" check box, the value entered will be added to any diffuse sources already present. If negative entries are made the diffuse source is reduced correspondingly. Existing diffuse sources can be deleted by entering a value **0**.

### 7.5.1    "Info" menu item

You will see information on the current system with the number of elements, nodes and boundary conditions in a dialog box.

### 7.5.2    "Project identification" menu item

You can enter a description of the current system; this will then be used in the general legend (see Section 7.6.10).

### 7.5.3    "Model" menu item

After going to this menu item or, alternatively, by pressing [**F5**], the program models the specified system using the data provided The following dialog box appears first:
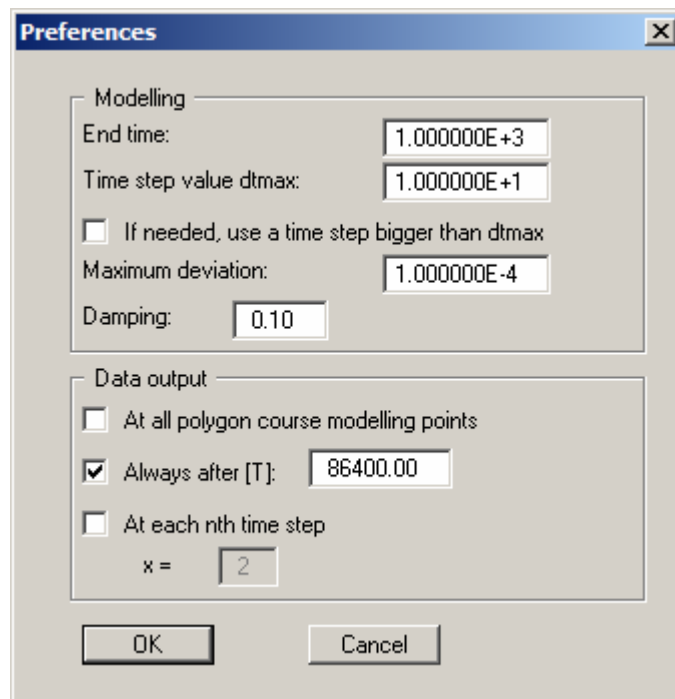


The following check boxes and/or preferences can be used in the dialog box:

- You define the end time ("**End time**") for the transient modelling (dimension as for permeability).

- You define the time increment ("**Time step value dtmax**") of the transient modelling (dimension as for permeability). The program also includes a time increment control. If you have selected a time increment that is too large, the time increment will be halved during transient modelling until sufficient precision is achieved. If you have selected a very small time increment, the program is also capable of adjusting the increment by doubling. If this is not wished for because, e.g., you need values at certain times for later evaluation, simply deactivate the "**If needed, use a time step bigger than dtmax**" check box.

- Iteration is necessary for every time increment. If the deviation of the potentials between the iteration steps (i) and (i - 1) is smaller than the "**Maximum deviation**", the program will jump to the next time increment. Maximum deviation values of 0.0001 are generally sufficient.

- You have the option of specifying a "**Damping**" value. During phreatic line modelling in complicated groundwater systems, oscillation around the actual solution can occur. This oscillation is damped by the damping factor. Values between 0.0 and 0.95 make the most sense. A value of 1.0 (complete damping) is nonsense and will therefore not be accepted by the program.

- A flood of results is usually the consequence of transient modelling, with correspondingly large result files. Often, a small time increment must be selected in order to achieve sufficiently precise results. You can therefore specify in the "**Data output**" group box the times at which data output should follow. Independently of the selected time increment, the program automatically proceeds to all times associated with boundary polygons.

  If you activate the "**At all polygon course modelling points**" check box, the results for all times will be given additionally to the usual input. So data output can either be at all polygon course modelling points, at a user-definable time [T], or at each desired ($n^{th}$) time increment. These three options can also be combined with one another. It is conceivable, for example, to enter a very large value for data output "**At each nth time step**" and to temporally subdivide a boundary condition polygon such that output only occurs at the user-defined times.

After entering your modelling preferences close the above dialog box using the "**OK**" button. The system information will then be displayed first (see Section 7.5.1) The modelling results are saved in a "**.plw3d**" file. The complete path can be viewed after clicking the "**Show complete file name**" button. Modelling starts after closing the message box by pressing "**OK**".

During modelling, a dialog box appears with information on the modelling progress. It is also provided with a "**Cancel**" button. If you press this button, modelling will not cease immediately, but instead you can edit specific program parameters such as maximum deviation, time step, etc. and then carry on modelling. If you bring modelling to an end the current time step is completed when you click "**Safe cancel**". You are provided with information on the modelling time and the size of the result file once modelling is complete.

### 7.6.1    "Refresh and zoom" menu item

The program works on the principle of *What you see is what you get*. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the screen is not refreshed after every alteration.

If, e.g., after using the zoom function (see below), only part of the image is visible, you can achieve a complete view using this menu item.
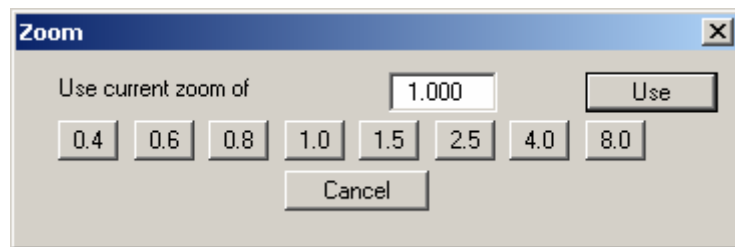


A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "**Use**" to exit the box the current factor is accepted. By clicking on the "**0.4**", "**0.6**", etc. buttons, the selected factor is used directly and the dialog box closed.

It is much simpler, however, to get a complete overview using [**Esc**]. Pressing [**Esc**] allows a complete screen presentation using the zoom factor specified in this menu item. The [**F2**] key allows screen refreshing without altering the coordinates and zoom factor.

### 7.6.2    "Zoom info" menu item

By clicking two diametrically opposed points you can enlarge a section of the screen in order to view details better. An information box provides information on activating the zoom function and on available options.

### 7.6.3    "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can edit the pen settings for various graphic elements (e.g. potential, sources, etc.). You can edit the pen widths for the elements shown in the dialog box; by clicking on the button with the element designation you can also edit the pen or fill colours.

On *monochrome printers* (e.g. laser printers), colours are shown in a corresponding grey scale. Graphic elements employing very light colours may be difficult to see. In such cases it makes sense to edit the colour preferences.

### 7.6.4    "Legend font selection" menu item

With this menu item you can switch to a different true-type font. All available true-type fonts are displayed in the dialog box.

### 7.6.5 "Draw Mini-CAD elements first" menu item

Objects added using **Mini-CAD** are normally drawn via the system representation. If the **Mini-CAD** objects are to be positioned behind the graphics this can be specified in the pop-up menu of the **Mini-CAD** module (layer dialog in **Mini-CAD**/"**Presentation**" button).

### 7.6.6 "Mini-CAD toolbar" and "Header toolbar" menu items

Using these two menu items you can add free text, lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.) to the main program graphics. PDF files can also be imported as images. The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual provided. The differences between the Mini-CAD and Header CAD are as follows:

- Objects created with **Mini-CAD** are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location of any foundations).

- Objects created with the **Header CAD** are based on the page format (in mm). This makes you independent of the coordinate system and keeps you in the same position on the page. You should select the "**Header toolbar**" if you wish to place general information on the drawing (company logo, report numbers, plan numbers, stamp etc.). Once you have saved the header information to disk (see **Mini-CAD** user manual), you can load it into completely different systems (with different system coordinates). The saved header information will appear in exactly the same position on the page, which greatly simplifies the creation of general page information.

### 7.6.7 "Toolbar preferences" menu item

After starting the program a horizontal toolbar containing selected menu items appears below the program menu bar. If you would rather work with a popup window with several columns, you can specify your preferences using this menu item. The smarticons can also be switched off.

At the bottom of the program window you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "**GGU-3D-TRANSIENT.ggu_alg**" file (see menu item "**Graphics preferences / Save graphics preferences**") and will be active at the next time the program is started.

Most of the program functions can be reached directly by clicking on the menu item tools (smart icons). If you hover with the mouse pointer over the tools the meaning of the smart icons appears in a text box. Some of the tool functions can be activated from the normal menu items.

 "**Zoom out**"
If you have previously *zoomed in*, this tool returns to a full screen display.

 "**Zoom (-)**" / "**Zoom (+)**"
With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.

 **"Colour on / off"**

If you need to remove the colour from the system presentation, to create a black and white print-out, for example, use this on / off switch.

 **"Copy/print area"**

Use this tool to copy only parts of the graphics in order to paste them, e.g. to a report. You will see information on this function and can then mark an area, which is copied to the clipboard or can be saved in a file. Alternatively you can send the marked area directly to your printer (see **"Tips and tricks"**, Section 8.3).

### 7.6.8 "3D toolbar" menu item

This menu item allows you to choose a pop-up window with tools for further editing the three-dimensional graphics. If you hover with the mouse pointer over the tools the meaning of the Smart icons appears in a text box.

The **"Rotation by …"** tool allows you to rotate the graphics around one of the three axes. The plus and minus signs designate the direction of rotation. The angle of rotation can be adjusted as wished using the  tool. The  tools allow you to move from one plane to the next. In addition, a number of menu items from the **"3D"** menu and the section definitions have been included in this toolbar.

### 7.6.9 "Layer/Plane legend" menu item

During input and graphical system evaluation you always find yourself in certain planes and layers. The number of the plane and layer are entered in a legend, if you have activated the "**Show legend**" check box in the dialog box for this menu item.
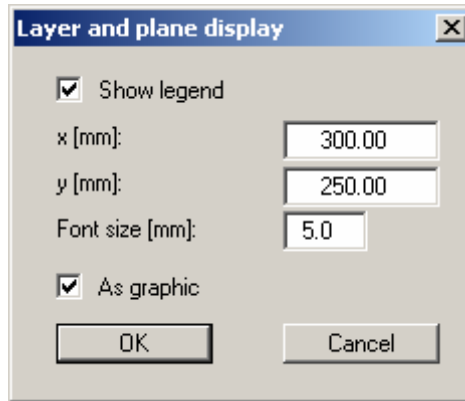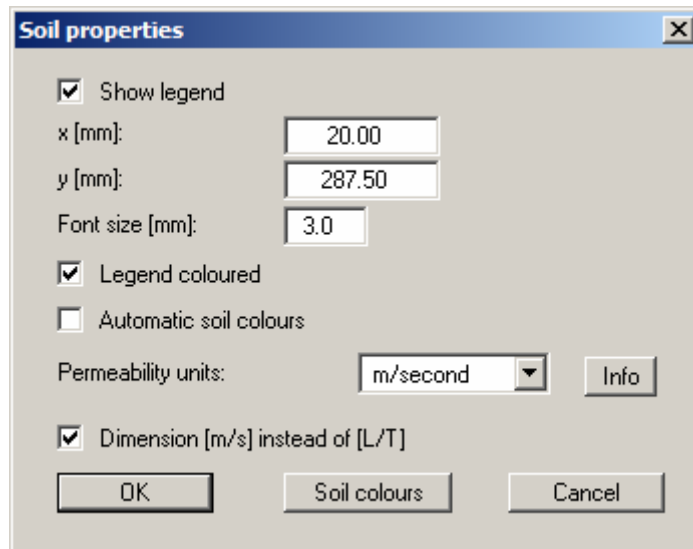


You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**".
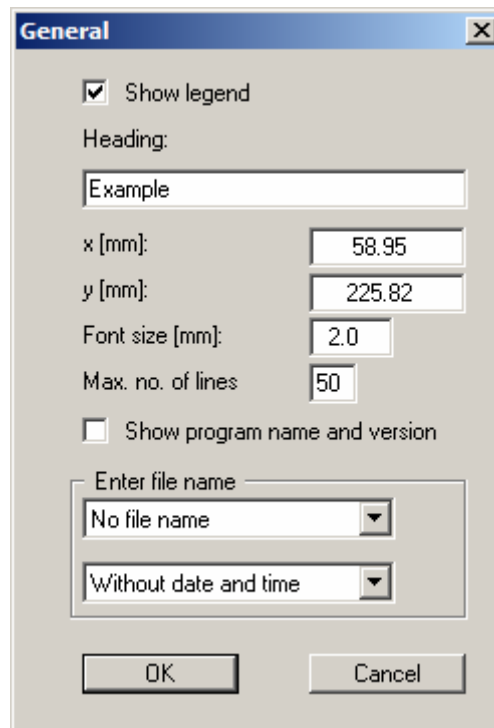
> The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position while holding the left mouse button.

If the "**As graphic**" check box is activated, the information will be displayed as a small graphical schematic as shown here.
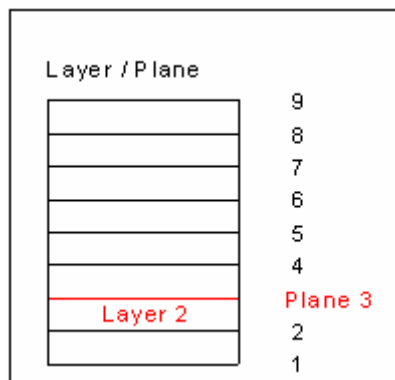
### 7.6.10 "General legend" menu item

A legend with general properties will be displayed on your output sheet if you have activated the
"**Show legend**" check box. Using this menu item you can alter the type of presentation.



In addition to the heading, this legend contains information on the current graphics and prefer-
ences. For example, if you display the FEM mesh you will see information on the number of tri-
angles and nodes.

You can define and edit the position of the legend using the values "**x**" and "**y**". You control the
size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns
are used.

> The fastest way to modify the position of the legend is to press the [**F11**] function key and
> then to pull the legend to the new position with the left mouse button pressed.

In the general legend you can, if wished, display information on the program (name and version)
and on the current file. Any project identification entered (see Section 7.5.2) will also be shown in
the general legend.

### 7.6.11 "Soil properties legend" menu item

A legend with the soil properties will be displayed on your output sheet if you have clicked the "**Show legend**" check box. Using this menu item you can alter the type of presentation or turn off the legend completely.



You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**".

> The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position while holding the left mouse button.

- "**Legend coloured**"
  The soil colours are displayed in the legend. Otherwise, they will be numbered.

- "**Automatic soil colours**"
  The soils are assigned soils colours automatically by the program. If the check box is <u>not</u> selected, the soil colours individually defined using the "**Soil colours**" button will be adopted.

- "**Soil colours**"
  You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each soil layer a new number or reorganise using the "**Soil colours/Reorganise**" command button. You can save your colour preferences to a file with "**Soil colours/Save**" and use them for different systems by means of the "**Soil colours/Load**" command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the "**Info**" button.

- "**Permeability units**"
  It often makes more sense in a transient analysis to enter the permeability in "**m/hour**" or "**m/day**". To avoid unnecessary explanations when submitting your analyses, the k-value is shown in the soil properties legend in "**m/second**". To achieve this, the program internally converts the entered coefficient of permeability from your selected unit back to m/s (see "**Info**" button).

- **"Dimension [m/s] instead of [L/T]"**
  The program uses [m/s] as the default unit. As explained above, you can also convert to [m/h] and [m/d]. If different units are required (e.g. yards and minutes), the general format length [L] and time [T] can be adopted. Just deactivate this check box.

### 7.6.12 "Section legend" menu item

If you activate the "**Show legend**" check box in the dialog box for this menu item, a legend containing a sketch showing the location of the section is displayed on the screen. The position of the legend can be defined and edited using the values "**x**" and "**y**", "**Width**" and "**Height**". The font size can be specified for the heading and labelling of the points.

### 7.6.13 "kr = f(u) function legend" menu item

The value kr is a dimensionless coefficient, representing the change in permeability in the unsaturated zone. It is specified in the **GGU-3D-SSFLOW** program when defining the soil properties for the initial steady-state system and used to model the phreatic line, among other things.

If you activate the "**Show legend**" check box in this menu item, the function is displayed as a graphic in a legend on the screen. The presentation can be modified to suit your needs in the dialog box.
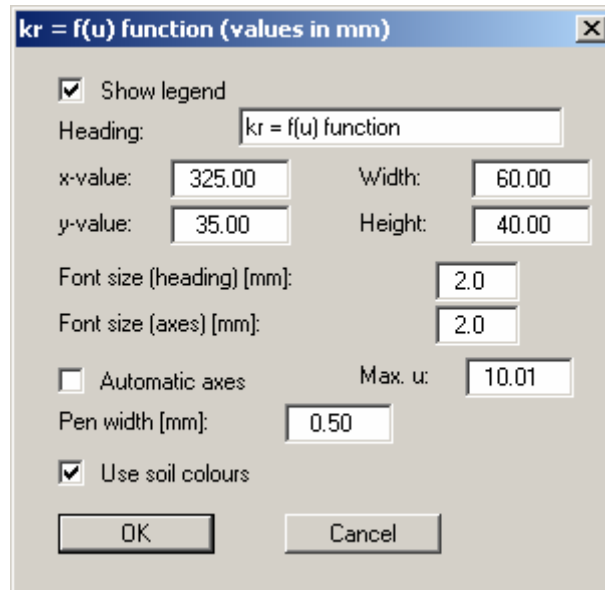


The position of the legend can be defined or edited by means of the variables "*x*" and "*y*", "**Width**" and "**Height**". If you edit the font sizes you should also always increase the width and height correspondingly, otherwise the actual function graphic will appear smaller within the defined legend frame.

If you activate the "**Automatic axes**" check box, the program will use a presentation scale including all given values for *u*. If the check box is deactivated, visualisation of the x-axis can be restricted to a user-defined "**Max. u**" value.

If the "**Use soil colours**" check box is activated, the individual graphs are colour-differentiated. However, if the graph values are the same for all soils, the function graph is automatically drawn in black and a corresponding note added to the graphics

### 7.6.14 "nw = f(u) function legend" menu item

The value of $n_w$ gives the volume of water relative to the total volume. It is a dimensionless quantity and dependent upon the pore water pressure u. The legend shows the specific characteristic curves of the user-defined soils.

In the dialog box to this menu item preferences for the visualisation can be edited if the "**Show legend**" check box is activated (see menu item "**Graphics preferences/kr = f(u) function legend**", Section 7.6.13).

### 7.6.15 "k = f(t) function legend" menu item

If you have entered different permeabilities as a function of time for the respective soil layers in the "**Time BC/Soil properties**" menu item, the function k = f(t) is displayed if the "**Show legend**" check box is activated.

In the dialog box to this menu item preferences for the visualisation can be edited (see menu item "**Graphics preferences/kr = f(u) function legend**", Section 7.6.13).

### 7.6.16 "Hydrograph legend" menu item

If the "**Show legend**" check box is activated in the dialog box, the lines used for the hydrographs are displayed in a legend, together with the corresponding node numbers.
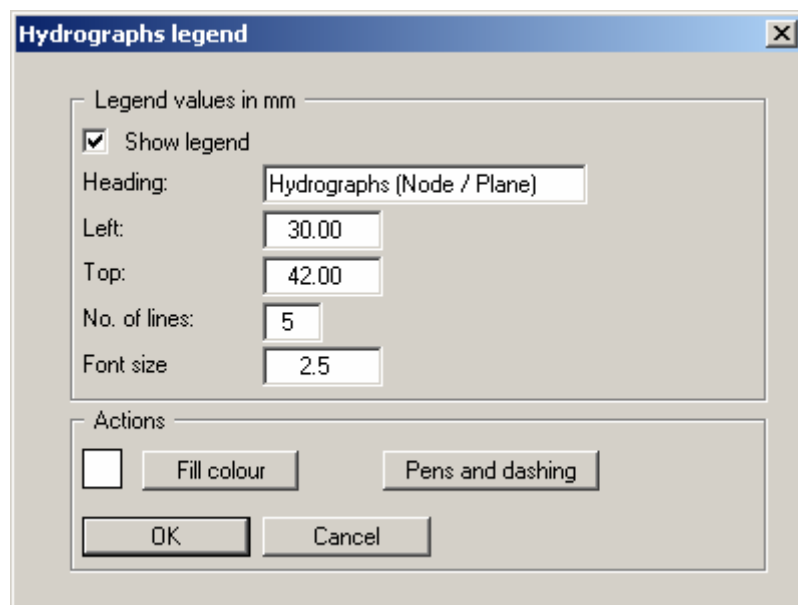
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used. The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

The heading given is automatically supplemented by the program with the state variable selected for hydrograph visualisation. The "**Pens and dashing**" button allows the individual hydrographs to be edited further.

### 7.6.17    "Hydrograph progress legend" menu item

This legend can be displayed on the screen if the potentials, velocities or discharges are evaluated following visualisation of the hydrographs (see the "**Times**" menu).
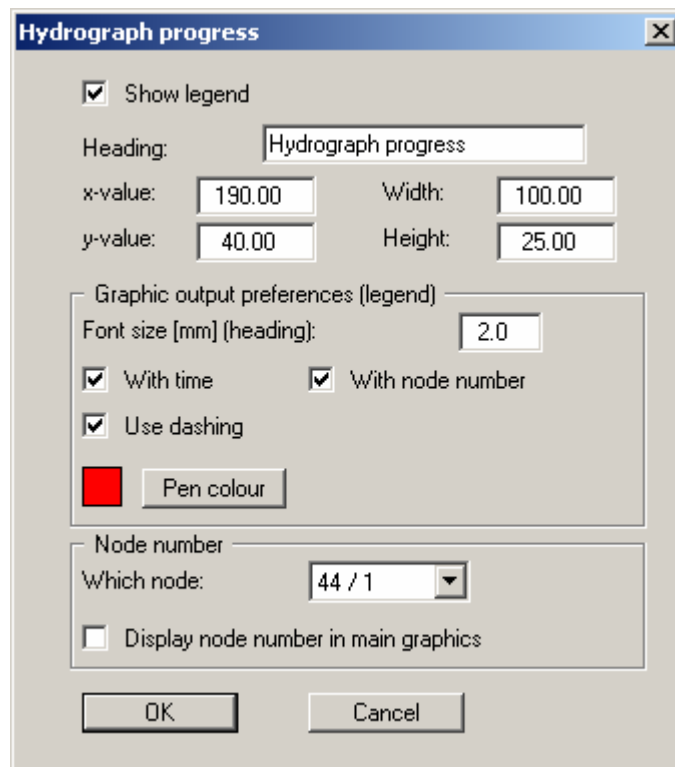


Activate the "**Show legend**" check box and select the node from the lower group box of the dialog box for which the hydrograph is to be displayed. If, for example, the potentials are now visualised as colour-filled contour lines, the respective point in time of the visualisation or the sequence of hydrograph time steps in the "**Flip book**" are marked in the legend.

### 7.6.18    "Move objects" menu item

Select this menu item in order to position legends or other graphical elements at the desired position on the output sheet. You can also move objects by pressing [**F11**] and then positioning the legend box with the left mouse button pressed. In that case an info-box appears no more.

### 7.6.19 "Save graphics preferences" menu item

Some of the preferences you made with the menu items of the "**Graphics preferences**" menu can be saved to a file. If you select "**GGU-3D-TRANSIENT.ggu_alg**" as file name, and save the file on the same level as the program, the data will be automatically loaded the next time the program is started and need not be entered again.

> If you do not go to "**File/New**" upon starting the program, but open a previously saved file instead, the preferences used at the time of saving are shown. If subsequent changes in the general preferences are to be used for existing files, these preferences must be imported using the menu item "**Graphics preferences/Load graphics preferences**".

### 7.6.20 "Load graphics preferences" menu item

You can reload a graphics preferences file into the program, which was saved using the "**Graphics preferences/Save graphics preferences**" menu item. Only the corresponding data will be refreshed.

## 7.7 Page size + margins menu

### 7.7.1 "Auto-resize" menu item

This menu item provides a to-scale visualisation, in both x and y coordinates, of the system and result graphics. If you have previously altered the image coordinates graphically or via editor, you can quickly achieve a complete view using this menu item. This function can also be accessed using the [**F9**] function key.

### 7.7.2 "Manual resize (mouse)" menu item

You can use the coordinates of a section of the visualisation as the new image coordinates by marking the desired area with the mouse, pressing the left mouse button and holding the [**Ctrl**] and [**Shift**] keys. The scales of the x- and y-axes are adjusted accordingly. If the previous proportions (scale x-direction/scale y-direction) need to be retained, the "**Proportional section**" check box must be activated.

Alternatively, you can simply redefine *origin* of the visualisation. The previous scale preferences are not affected by this.

### 7.7.3 "Manual resize (editor)" menu item

You can alter the image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the *drawing area*. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 7.7.4).

The image coordinates entered here can be saved in a file with the extension "**.ggu-bxy**" and be reloaded later for the same file or for different files.

If you want to recover the previous values during input or use the menu item again after editing the coordinates, you can do this by pressing the "**Old values**" button.

### 7.7.4 "Page size and margins" menu item

The default page set-up is A3 when the program is started. You can edit the page format in the following dialog box.



- "**Page in general**" defines the size of the output sheet. The A3 format is set as default. The program automatically draws thin cutting borders around the page, which are required when using a plotter on paper rolls. The borders can be switched off using the "**With borders**" check box in the "**Page size + margins/Margins and borders**" menu item (see Section 7.7.6).

- "**Page margin**" defines the position of a frame as a distance to the margins. This frame encloses the subsequent diagram. You can switch off the frame deactivating the "**With margins**" check box in the "**Page size + margins/Margins and borders**" menu item (see Section 7.7.6).

- The "**Plot margin**" define a set distance between the page margin and the actual *drawing area* in which the graphical evaluation of your input is presented.

### 7.7.5 "Font size selection" menu item

You can edit font sizes for labelling the various drawing elements.

The font sizes of text within legends are edited in the respective legend editor. Just double-click in a legend to do this.

### 7.7.6 "Margins and borders" menu item

The program automatically draws thin cutting edges around the page, which are required when using a plotter on paper rolls. Page margins (see menu item "**Page size + margins/Page size and margins**") defines the position of a frame as a distance to the cutting border. This frame encloses the subsequent diagram. You can switch off the lines by deactivating the "**With margins**" and "**With borders**" check boxes.

### 7.8.1    "Read next time step" menu item

By clicking this menu item a system image is generated displaying the flow conditions for the next time step. The time step is defined beforehand for modelling or for output in **GGU-3D-TRANISENT**.

This function can also be called by pressing [**F6**] or by clicking the [↑] icon in the menu toolbar.

### 7.8.2    "Read previous" menu item

A system image of the flow conditions for the previous time step is generated The time step is defined beforehand during modelling.

This function can also be called by pressing [**F7**] or by clicking the [↓] icon in the menu toolbar.

### 7.8.3    "Read specified" menu item

With the help of this function the system can be displayed on the screen for a user-defined time step **X** and, if required, be sent to the printer. This function can also be called by clicking the [↕] icon on the menu toolbar.

### 7.8.4    "Flip book" menu item

This is one of the program's most impressive methods of evaluation. One of the menu items for contour lines or velocity visualisation from the "**Evaluation**" menu must have been previously started. Then go to this menu item to open the following dialog box:
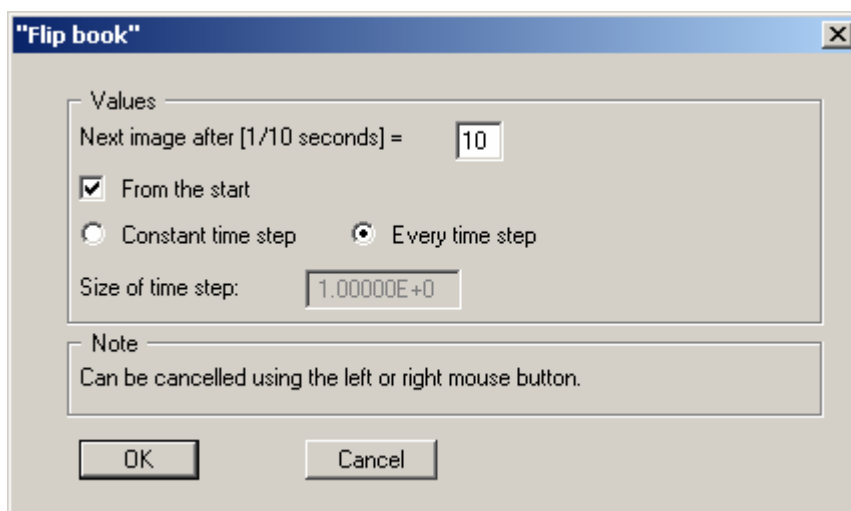


Here, you specify the speed of the animation, i.e. after how long a new image is to be displayed on the screen. Finally, you specify the flow time after which the groundwater situation is to be displayed. Using the "**OK**" button, you can start the animation. If you would like to abort before the end time is reached, press any mouse button.

---

### 7.8.5 "Maximum values for all times" menu item

You will be given information on the maximum values of potential, velocity and discharge determined during modelling.

### 7.8.6 "Time format" menu item

Here, you specify the number of decimal places for the time data. If the "**Time as date + time**" check box is activated, the start time and time units for the subsequent visualisation can be specified.
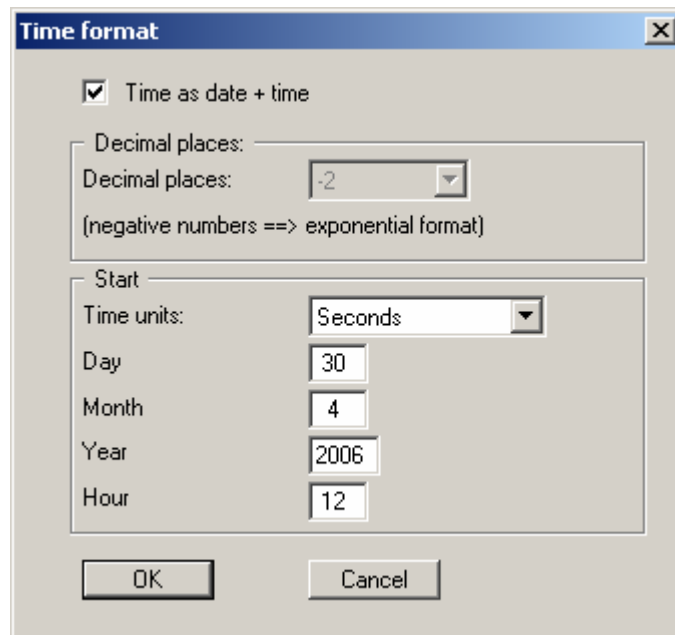


### 7.8.7 "Define hydrograph points" menu item

By clicking selected node points with the left mouse button the program will evaluate the temporal development of one of the flow variables for these points. After closing the message box you are first asked which flow variable should be evaluated for these points. You may select between the potential, the seepage velocity in x and y directions or the discharge for screen display.

If you want to evaluate flow variables as hydrographs for further nodes, click the "**Keep current nodes**" option button before closing the dialog box. After clicking "**Define new nodes**" the existing nodes are deleted from the list.

### 7.8.8 "Display hydrographs" menu item

Once the time values have been read in, you will be asked which values you would like to have displayed at given points. The list of points with the associated, defined flow quantities help to simplify the selection. If you have assigned the same flow value to all node points, you can have the temporal development displayed for all activated nodes simultaneously by clicking the "**All**" button. This can be very advantageous if you would like to compare the changes at different regions in the system. The following dialog box will be subsequently displayed:
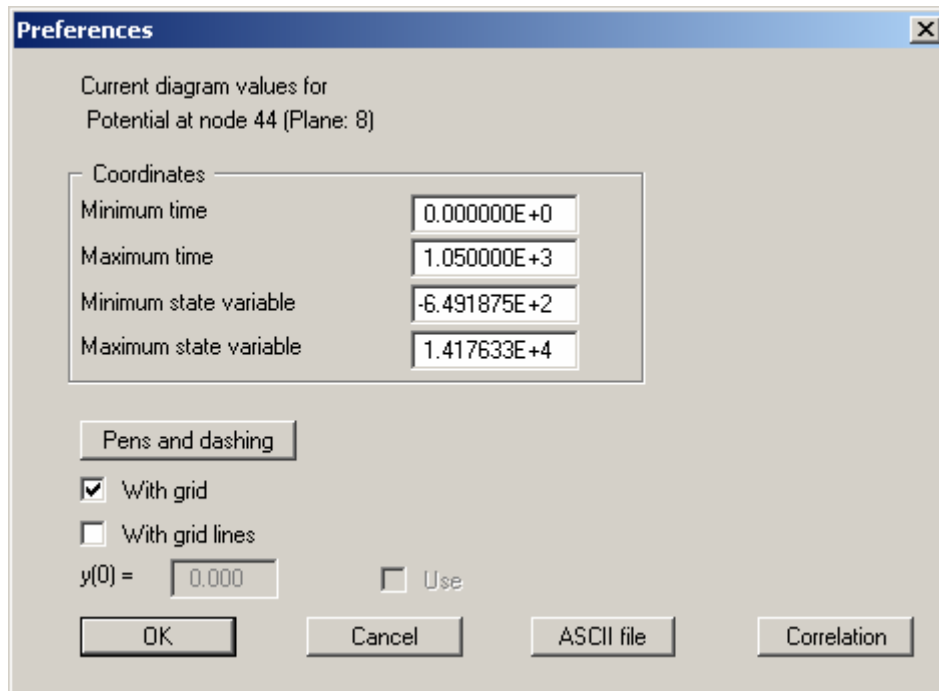


The box shows the minimum and maximum values at the node points. Using the "**ASCII file**" button, the data can be saved in a record that can be represented as a table, for example, in your word processor.

### 7.8.9 "Display position" menu item

Using this menu item, the clicked or activated node points for which the flow variables are to be visualised as hydrographs are displayed in the system.

### 7.8.10 "Save nodes" menu item

After selecting this menu item you can save defined hydrograph points in order to have them available again at a later date. For reasons of clarity, it makes sense to use "**.knt_3d**" as file suffix, as this is the suffix used in the file requester box for the menu item "**Times/Load nodes**". If you choose not to enter an extension when saving, "**.knt_3d**" will be used automatically.

### 7.8.11 "Load nodes" menu item

After selecting this menu item, previously defined system hydrograph points can be opened using normal WINDOWS operations.

---

### 7.9.1     General notes

The "**Evaluation**" menu, among others, is provided for graphical evaluation of the FEM model. The results are displayed for the currently active area (selected plane or activated section). If the area being considered here is a plane, you can move back and forwards between the planes by clicking the arrow buttons on the *3D toolbar* (see Section 7.6.8). The respective current layer and plane are shown in the *status bar* at the bottom of the window (see Section 7.6.7) or in the *layer/plane legend*, if activated (see Section 7.6.9).

If you are currently within a system section view, the potential contour lines, discharges or velocities are shown in the section. If you are within the system section view and want to view in the node plane, first go to the "**Mesh(x/y)/Mesh**" menu item (see Section 7.2.2).

The currently displayed time step is shown in the *status bar*. You can move between time steps via the corresponding menu items in the "**Times**" menu (see Sections 7.8.1 to 7.8.3) or directly by using the arrow tools in the *menu items toolbar*.

### 7.9.2     "Normal contours" menu item

You can visualise lines of equal water level (potentials) using this menu item. You will see the following dialog box, in which you can specify the type of presentation.
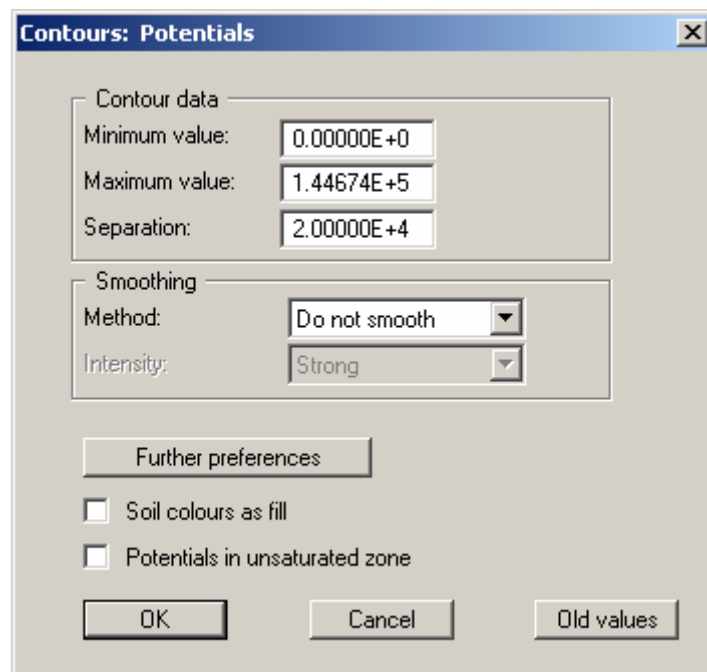


The program shows you the existing smallest and largest values and the spacing of the contour lines. If you want the visualisation to begin at a different value, the initial value can be entered here. You can also vary the spacing, for example, in order to reduce the number of contour lines drawn.

When you select this menu item the settings displayed here are always those automatically selected by the program. Using the "**Old values**" button, the preferences used for the previous contour line diagram are adopted. This information is saved with the record.

You can also select from three smoothing out procedures:

- "**Do not smooth**"
  As a linear method is implemented within the triangle elements, this will provide modelling results without smoothing performed by the program.

- "**Method 1**" and "**Method 2**"
  these smoothing out procedures employ two different Bezier splines. Smoothing out intensities can be given for both in order to acquire smooth contour lines. **Method 2** creates very "**round**" contours, with the danger that the true results may be falsified.

You can specify the type of visualisation for the contour lines by means of the "**Further preferences**" button. By activating the appropriate check box the soil colour can be used as fill.

> As the program calculates with unsaturated permeabilities, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone with the corresponding switch.

### 7.9.3    "Coloured" menu item

Lines of equal water level (potentials) can be displayed as colour-filled contours using this menu item.
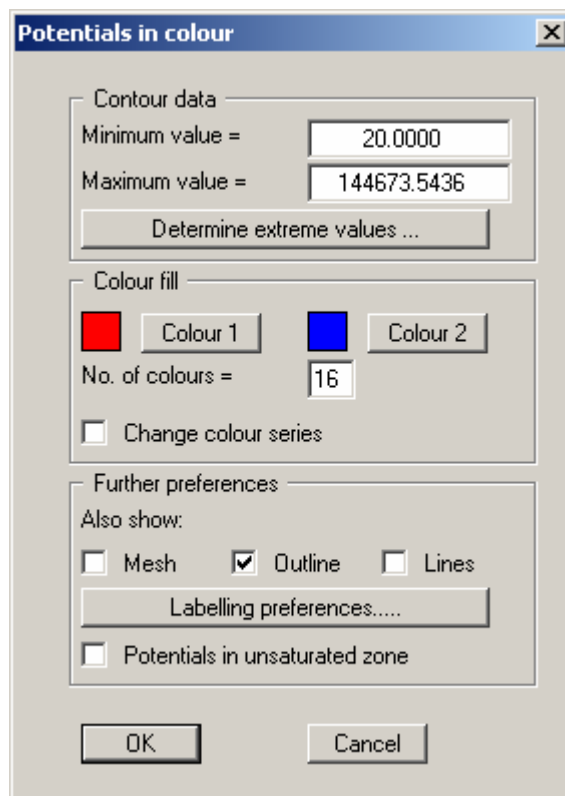
For colour-filled graphics you will see the above dialog box.

- "**Contour data**" group box
  Press the "**Determine extreme values…**" button. The program then determines the minimum and maximum values for the corresponding layer base. You can then edit these values, for example in order achieve a defined start value.

- "**Colour fill**" group box
  You can control the colour subdivisions of the contour diagram using "**No. of colours**". In the example above, 16 colours will be displayed between "**Colour 1**" and "**Colour 2**". The default setting is a colour course from red to blue. These colours can be edited as required after selecting the "**Colour 1**" and "**Colour 2**" buttons, or simply reverse the choice by selecting the "**Change colour series**" check box.

- "Further **preferences**" group box
  In addition to the colour presentation you can also have the triangle mesh and/or the outline displayed. Additional contour lines can also be drawn. Line labelling preferences can be defined by means of the "**Labelling preferences**" button.

- "**OK**"
  The colours will be drawn after confirmation.

A colour bar at the right edge of the sheet allows correlation between the colour and the corresponding value. If this colour bar is drawn in the right page margin, specify a larger value for the right plotting margin (e.g. 25 mm) in the "**Page size + margins/Page size and margins**" menu item (see Section 7.7.4).

---

As the program calculates with unsaturated permeabilities, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone with the corresponding switch.

---

### 7.9.4 "3D" menu item

Using this menu item it is possible to visualise lines of equal water level (potentials) forfor the existing nodes in three dimensions. In the following dialog box you can choose between perspective and parallel projection, as well as selecting the position of the *eye*.
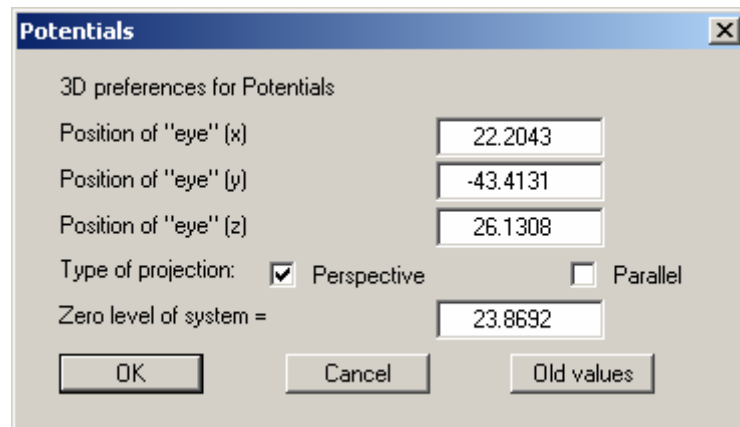


The program will generally make sensible suggestions for the input values in the dialog box, so you will not normally need to make alterations.

---

After leaving the dialog box by pressing "**OK**" you will see the following dialog box. It greatly resembles the box shown in the menu item "**Evaluation/Coloured**".
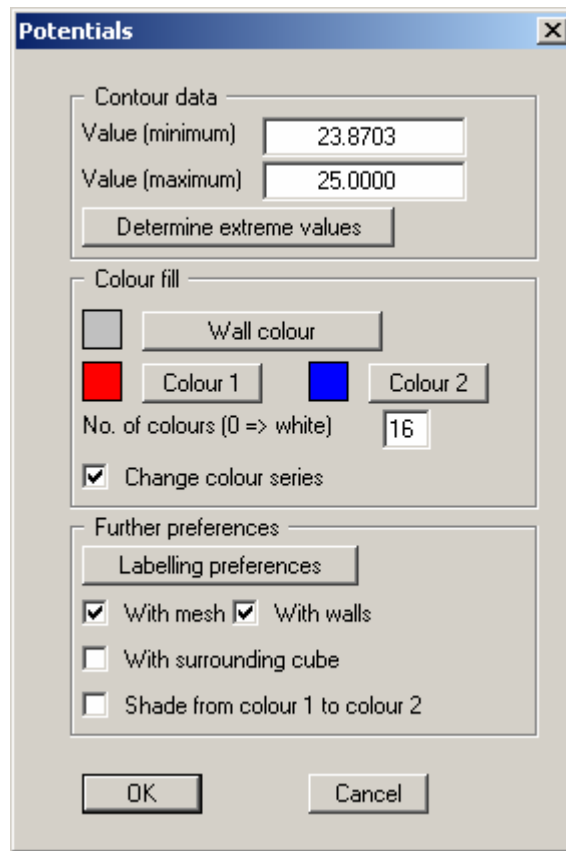


You can also specify whether an enclosing cube should be displayed; in some cases this improves the 3D effect. The "**Shade from colour 1 to colour 2**" check box represents a bit of a special effect. If this check box is activated a light source is simulated in the region of the *eye*. The angle between the light beam and the respective 3D surface represents a measure of the reflection. The area is shaded in accordance with colours 1 and 2. A good choice of colours, for example, would be dark grey for colour 1 and pale grey for colour 2.

### 7.9.5 "3D array" menu item

The 3D effect is usually lost in an irregular triangular mesh. The "**Evaluation/3D array**" menu item is useful here. After selecting the required parameter the following dialog box opens where you can define the array.

**Potentials** ✕

3D graphics rectangular array for Potentials

xmin / xmax = 0.000 / 50.000

ymin / ymax = 0.000 / 25.000

Number of "plates" in x: `15`

Number of "plates" in y: `15`

[ OK ]      [ Cancel ]
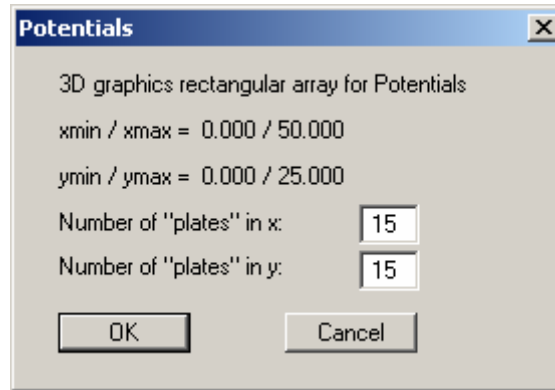
After leaving the dialog box by pressing "**OK**" you will at first see the same box as that described in "**Evaluation/3D**". The program calculates the state variables at the array points by linear interpolation of the results of the FEM analysis. You then see the dialog box for coloured contours preferences as described in "**Evaluation/3D**" (see Section 7.9.4).

### 7.9.6 "Discharges" menu item

The FEM mesh will be displayed at the nodes, together with the calculated discharge. Discharges other than zero are present at the nodes at which you have defined a water level boundary condition or a source boundary condition. At all other nodes the discharge equals zero (continuity condition). Using the dialog box you can specify the type of presentation.
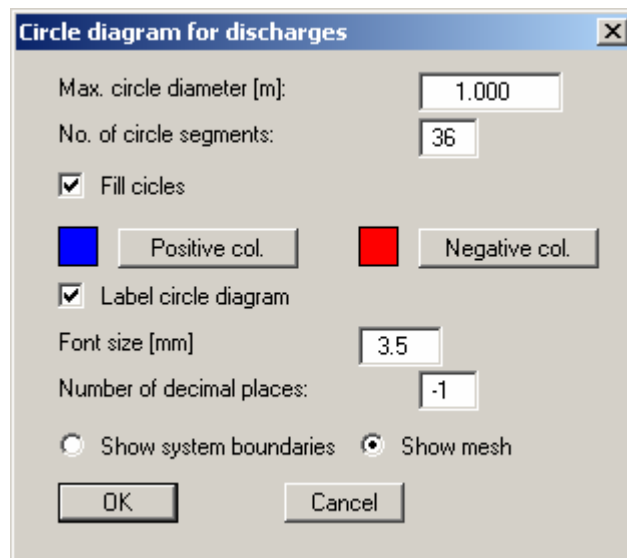
After leaving the box using "**OK**" the discharges are displayed as variable sized circles. Positive and negative discharges can be displayed in different colours in filled circles.

By specifying a appropriate areas of the screen, the sum of the discharges in these areas can be calculated by the program. By clicking any four points anti-clockwise the discharge in the enclosing polygon is determined for the selected plane and for all other planes and displayed in a message box.
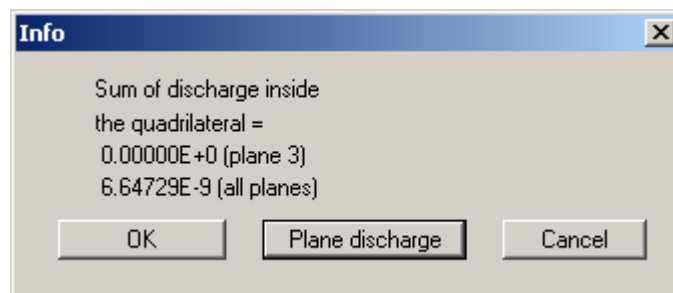
You can then calculate the plane discharge for the selected area. The plane discharge is the discharge flowing through the triangles contained within the quadrilateral of the current plane. It is calculated from the node discharges of the triangular prisms located below the plane. Under certain circumstances, the boundary nodes may include a component belonging to a neighbouring prism not included within the quadrilateral, leading to slight inaccuracy.

### 7.9.7 "Discharge in vertical section" menu item

After defining a section course, the total discharge in the defined section and the partial discharges from the current to the uppermost plane are computed. The result is presented in a message box. Here, you can opt to save the current section in a "**.ggu_fes**" file.

---

### 7.9.8 "Values along line" menu item

You can define a section using the left mouse button by clicking the appropriate nodes. Intermediate points are automatically supplemented by the program. A click with the right mouse button deletes the last point from the list. When the section is located where you want it press [**Return**]. You will see a dialog box where you can define preferences for the type of visualisation.
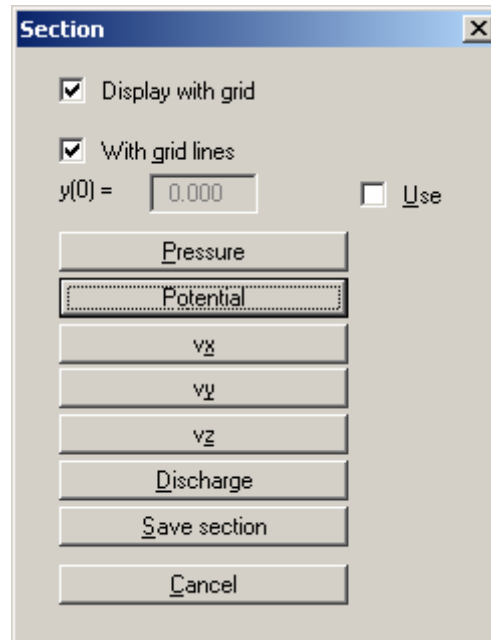


After clicking the parameter to be visualised the values along the defined section will be displayed graphically. You can save the previously defined section to a file. It can then be opened again if you return to this menu item. If you used this menu item immediately preceding this point, you can also open the last section using the "**Prev. section**" button.

### 7.9.9 "Velocities + gradients" menu item

The velocity field can be visualised using this menu item. The size and direction of the seepage velocity are shown ($v_s = (k \cdot i)/n_{eff}$!).
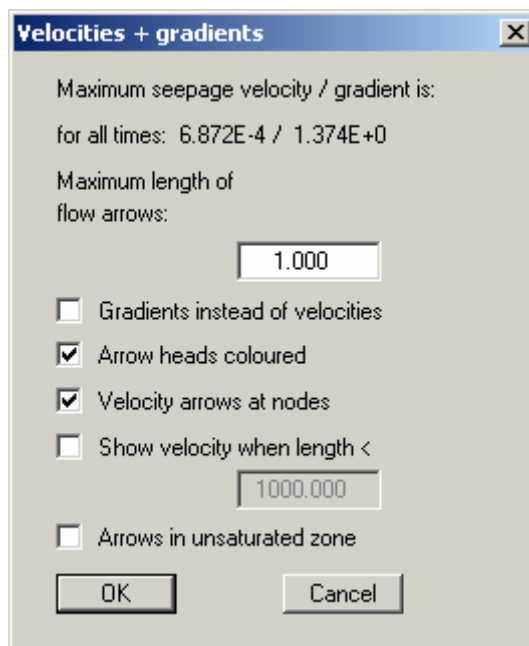


First, enter the length of the maximum seepage velocity to be displayed. You can then decide whether the velocities or the gradients are to be displayed.

By selecting the "**Velocity arrows at nodes**" check box, you can determine whether or not the velocities are to be displayed at the nodes or at the centroids of the triangular elements. The velocity at the centroid is determined by the program from the FEM analysis. The velocities at the element nodes are determined as a mean of the *centroid velocities* of the neighbouring triangular elements. The length of the maximum velocity vector must be given in the user-defined length unit (generally metres). In the following presentation you will see scale information for length/time units at the top left. The colour of the arrowheads can be edited using the "**Graphics preferences/Pen colour and width**" menu item (see Section 7.6.3).

For systems with, e.g., wells, very high velocities occur in the area of the well, which may possibly *ruin* the graphics. Select a length for which velocities should still be presented and activate the corresponding switch.

> As the program calculates with unsaturated permeabilities, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone with the corresponding switch.

### 7.9.10 "Potentials + velocities" menu item

Potentials and flow velocities can also be displayed simultaneously, thus illustrating their interdependency The dialog boxes for contour line and velocity display preferences are shown in sequence. The dialog box descriptions can be found in Sections 7.9.2 und 7.9.9.

### 7.9.11 "Maximum gradients" menu item

You will see the determined maximum gradients for each soil displayed in a message box.

```
Maximum gradients                                    [x]

  Time =  1.0E-6

  Soil 1 i(max) = 0.00000 x/y = -12.588 / 3.659 (Layer  17)
  Soil 2 i(max) = 134.48688 x/y = -19.599 / 6.003 (Layer  11)
  Soil 3 i(max) = 40.77759 x/y = -19.566 / 2.018 (Layer  2)
  Soil 4 i(max) = 40.77759 x/y = -19.566 / 2.018 (Layer  1)
  Soil 5 i(max) = 0.00000 x/y = -1.389 / 2.002 (Layer  7)
  Soil 6 i(max) = 0.00000 x/y = 8.503 / 3.650 (Layer  7)
  Soil 7 i(max) = 0.00000 x/y = 16.089 / 5.996 (Layer  17)

     [    OK    ]           [   Copy to clipboard   ]
```
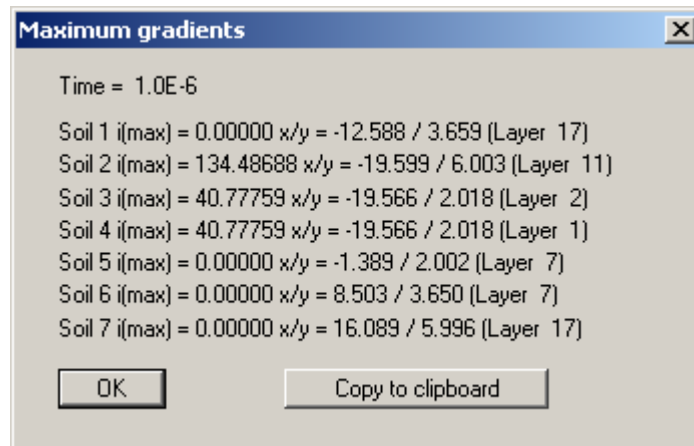
If you want to use the values e.g. in your report, you can import them to your word processing program by clicking on the "**Copy to clipboard**" button.

### 7.9.12 "Individual values" menu item

By left-clicking an FEM node all modelling results for the respective node in the current plane can be viewed. If you need to view the values for the same node in a different plane, first move to the plane required using the arrow buttons in the *3D toolbar* and click the node again. If you have defined a section you can directly click the nodes in the various planes.

### 7.9.13 "Define section" menu item

Define the required section by clicking system nodes with the left mouse button. The node points are numbered. Intermediate points are automatically generated by the program. The last point defined is deleted from the list by clicking the right mouse button. You can also define a *circular section*, ending back at the starting point. If the circular section is defined anti-clockwise, it protrudes like a tower. If the circular section is defined clockwise, you will see a *depression;* you can "see inside" this "hole". When the section is how you want it press [**Return**].

On the screen you will then see the FEM mesh for your defined section. In addition, a site plan is automatically displayed on the screen, showing the system outline and the defined section. This site plan can be switched off using the menu item "**Graphics preferences/Section legend**" by deactivating the "**Show legend**" check box (see Section □).

In the "**Evaluation**" menu you can use one of the numerous options for graphical visualisation of the defined section, for example as potential contour lines or discharges. The respective time step is displayed in the status bar. In order to move from the section view to the normal system view for evaluation, go to the menu item "**Mesh(x/y)/Mesh**" (see Section 7.2.2).

If you have previously saved a section it can be loaded again using "**Load section**". The "**Prev. section**" button is only visible if a section has already been defined and loads the section defined immediately prior.

### 7.9.14    "Mesh (section)" menu item

When you select this menu item the FEM mesh for the defined section is displayed.

### 7.9.15    "Outline (section)" menu item

When you select this menu item the system outline for the defined section is displayed.

### 7.9.16    "Fence diagram" menu item

Using this menu item you can create a fence diagram for computed quantities in a section. The section is defined using the "**Evaluation/Define section**" menu item.
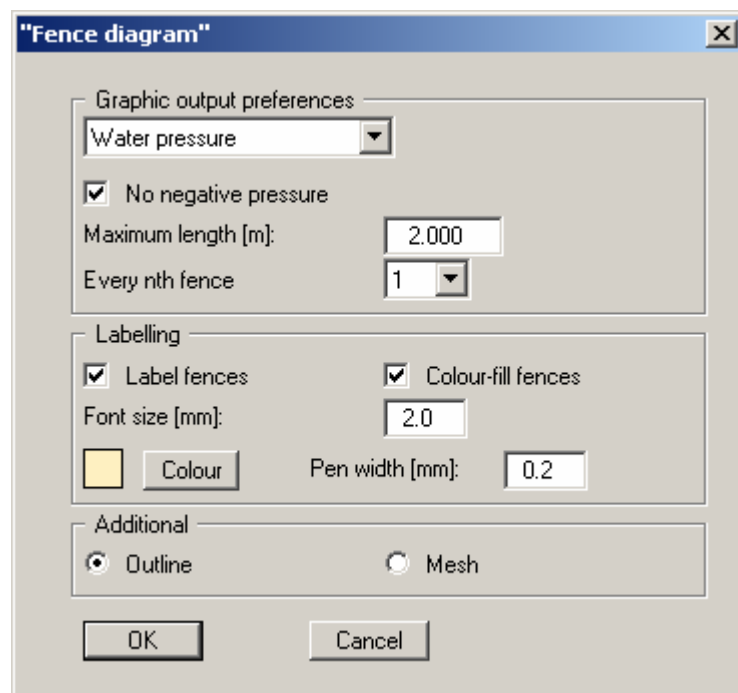


In the dialog box, choose between visualisation of the water pressure or the potential. You can then specify preferences in accordance with the selected quantity.

### 7.10.1   "Wire model" menu item

After specifying the type of projection and visualisation, the system is displayed as a wire model, i.e. all nodes are connected by lines. To improve clarity, visualisation of the plane nodes can be restricted to the top and bottom plane by activating the "**Top and bottom only**" check box.

### 7.10.2   "External model" menu item

You can display an external view of the model. The surfaces are displayed using the soil type colours defined in "**Graphics preferences/Soil properties legend**".
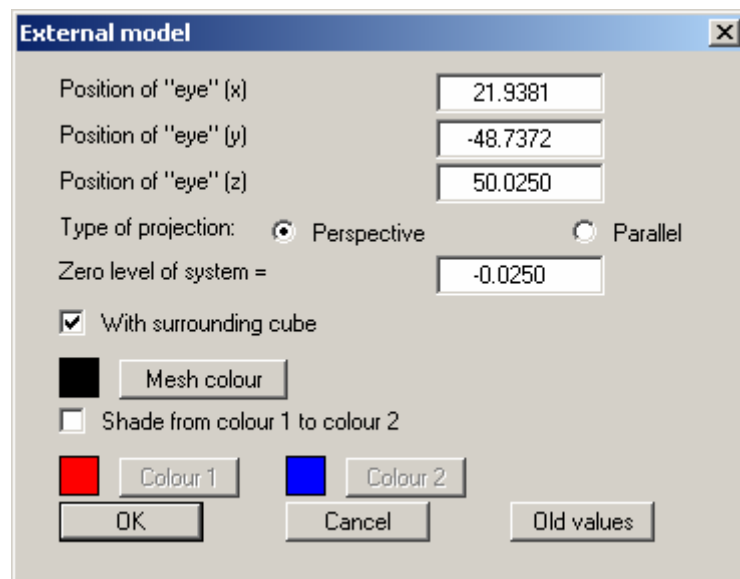


If you activate the "**Shade from colour 1 to colour 2**" check box, visualisation does not use the soil colours but instead uses shading in the two specified colours. If you previously defined a section, you will see a cut-away 3-dimensional system.

The outside of the system is covered by the mesh to increase the 3-dimensional impression. The colour of the mesh can be altered using the "**Mesh colour**" button.

After confirming your preferences by pressing "**OK**" the 3D model is displayed on the screen. The angle of view of the model can be altered using the rotation tools in the *3D toolbar*. You can view the original visualisation once more by returning to this menu item and clicking the "**Old values**" button. If the model leaves the screen while it is being rotated, press the [**F9**] function key once and the screen coordinates will be recalculated.

You can move between the planes of your system using the arrow buttons on the *3D toolbar*. However, this only makes sense for cut-away system visualisation (external model in section).

### 7.10.3 "Potential external model" menu item

You can display an external view of the model that includes the potentials. After defining the angle of view and the type of projection you will see a dialog box for specifying potential visualisation preferences.
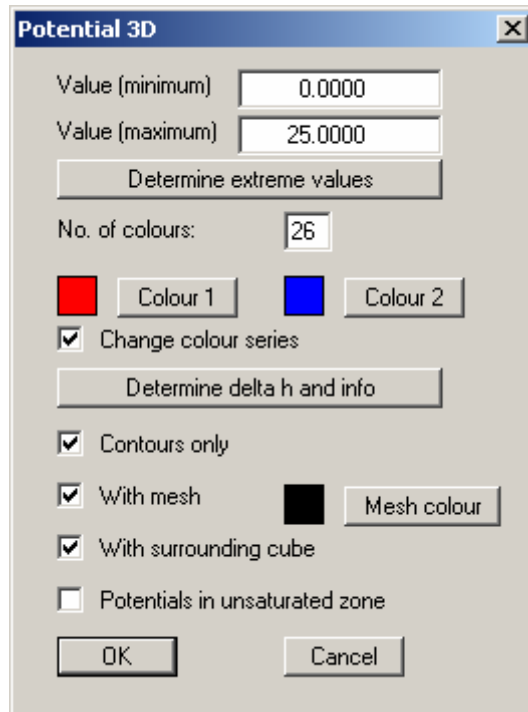


After determining the extreme values you define the boundary colours for the continuous colour flow using the "**Colour 1**" and "**Colour 2**" buttons. If you previously defined a section, you will see a cut-away 3-dimensional system, including the potentials on the section surfaces.

If you activate the "**Contours only**" check box, no colour fill will occur and only contour lines will be drawn. The contour lines are then allocated "**Colour 1**". The spacing of the contour lines is determined by "**Value (minimum)**" and "**Value (maximum)**" as well as the number of colours specified. To save bothersome calculations you can display the spacing beforehand using the "**Determine delta h and info**" button. If necessary, change the number of colours.

The outside of the system is covered by the mesh to increase the 3-dimensional impression. The colour of the mesh can be altered by means of the "**Mesh colour**" button.

After confirming your preferences using "**OK**" the 3D model is displayed on the screen. The angle of view of the model can be altered using the rotation tools in the *3D toolbar*. You can view the original visualisation once more by returning to this menu item and clicking the "**Old values**" button in the initial dialog box. If the model leaves the screen while it is being rotated, press the [**F9**] function key once and the screen coordinates will be recalculated.

You can move between the planes of your system using the arrow buttons on the *3D toolbar*. However, this only makes sense for cut-away system visualisation (external potential model in section).

### 7.10.4 "Potential surface" menu item

Surfaces with equal potentials, specified by user-input in "**Value of potential surface**", can be visualised in three dimensions in your system.
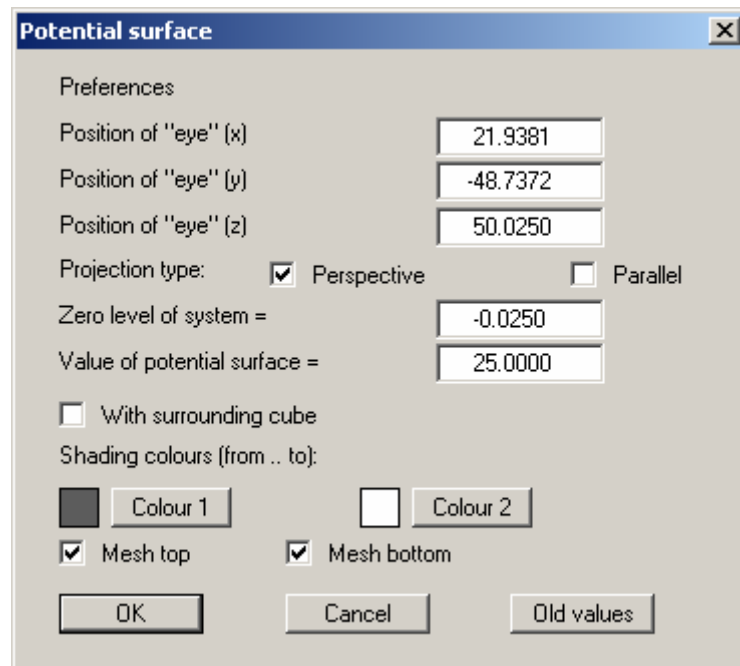


Shading corresponds to illumination of the system originating at the eye. The "**Flip book**" menu item in the "**Times**" menu is particularly impressive.

### 7.10.5 "Seepage surface" menu item

You can display your system's seepage surface in three dimensions. You will see a dialog box analogous to the box in the previous menu item (see Section 7.10.4). The "**Flip book**" menu item in the "**Times**" menu is particularly impressive.

### 7.10.6 "Flow lines preferences" menu item

#### 7.10.6.1 *Principles*

Flow lines can be drawn from the velocities calculated by the finite-element program. The principle used by the program is also known as the course line method or particle-tracking method. With this method a water particle, which wanders through the system from any start point, is observed. The program determines size and direction of seepage from this start point. With this velocity vector, the new position of the particle can be determined, whereby either a "**distance increment**" or a "**time increment**" can be selected for calculation of velocity vector length. The new position of the water particle is then the start point for the next calculation step. This procedure is carried on as long as a user-defined end-time is not exceeded, or the particle leaves the system. Small increments mean high precision and long calculation time.

If the increments are too large, nonsensical results can occur. Calculations are performed using the seepage velocity ($vs = (k \cdot i)/neff!$). Using this program, you can also follow a particle back (against flow direction).

#### 7.10.6.2 *Preferences*

After going to this menu item the flow modelling points are defined using the buttons in the following dialog box.
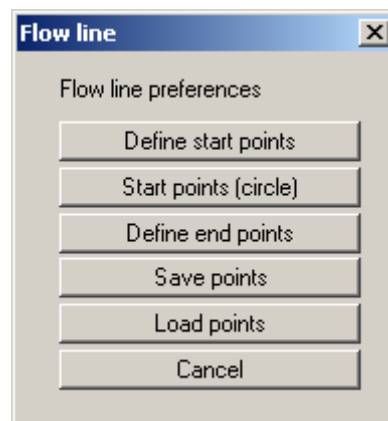


- "**Define start points**"
  You can define the start points of water particles.

- "**Start point (circle)**"
  You can define several start points, which can be grouped in a circle around the point you have determined. This can simplify the input of water particles, e.g., for wells within the system (key word: "**50 day line**").

- "**Define end points**"
  If depressions (extraction wells) are present within a system, the result may, in certain circumstances, be an oscillation of the particle track around the well, as all velocities are directed towards the well. In this case, end points can be defined in order to avoid unnecessarily long computation times. A square area must be defined by the user around the end points. If a water particle *enters* this square, computation of the corresponding particle track is ended.

- "**Save points**"
  You can save start points to a file, in order to have them available for a different system.

---

- **"Load points"**
  Start points saved using the previous button can be reloaded into the program using this button.

After selecting one of the switches for defining points, the following box opens:
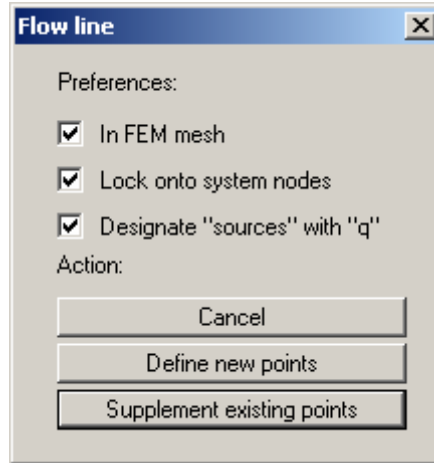


The "**In FEM mesh**" check box displays the FEM mesh when specifying points; otherwise, only the system outline is displayed. Defined points will be locked on to the system nodes closest to the mouse pointer when clicked, if the "**Lock onto system nodes**" check box is activated. Otherwise the points can be freely positioned within the system.

If some points are already defined, further points may be added by pressing the "**Supplement existing points**" button. Existing points are deleted if you exit the box via the "**Define new points**" button.

### 7.10.7 "Distance increment" menu item

You can start modelling the particle path(s) using a distance increment. You first define the angle of view and the type of projection in a dialog box. To improve clarity, visualisation of the 3D model can be restricted to the top and bottom plane by activating the "**Top and bottom only**" check box.

In the following dialog box the modelling parameters and the type of subsequent presentation can be confirmed or edited. Activate the "**Opposite to flow direction**" check box in order to retrace the path of a particle.
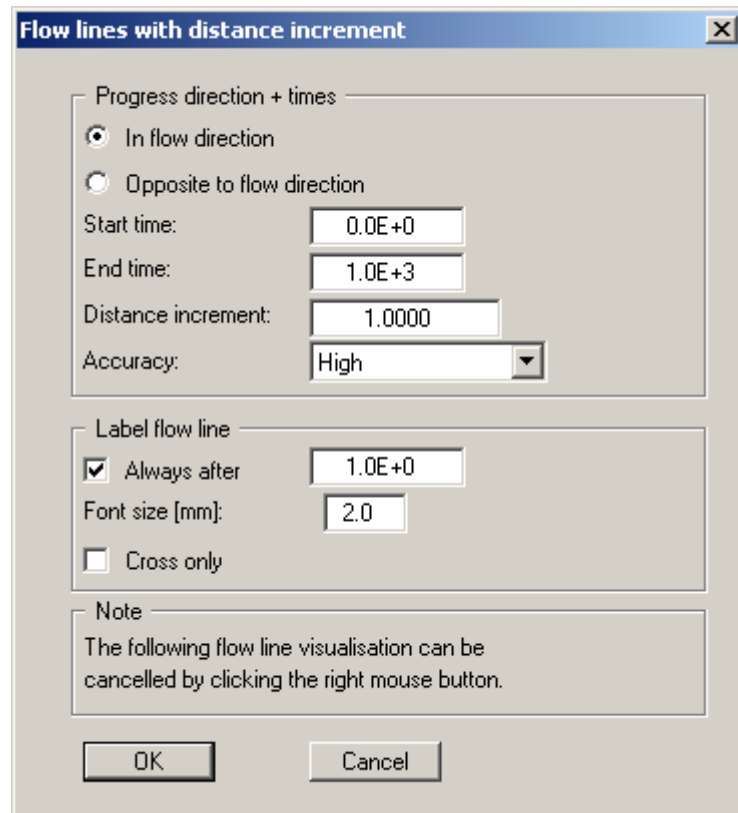
### 7.10.8 "Time increment" menu item

You can start modelling the particle path(s) using a time increment. You first define the angle of view and the type of projection in a dialog box. To improve clarity, visualisation of the 3D model can be restricted to the top and bottom plane by activating the "**Top and bottom only**" check box.

In the following dialog box the modelling parameters and the type of subsequent presentation can be confirmed or edited. Activate the "**Opposite to flow direction**" check box in order to retrace the path of a particle.
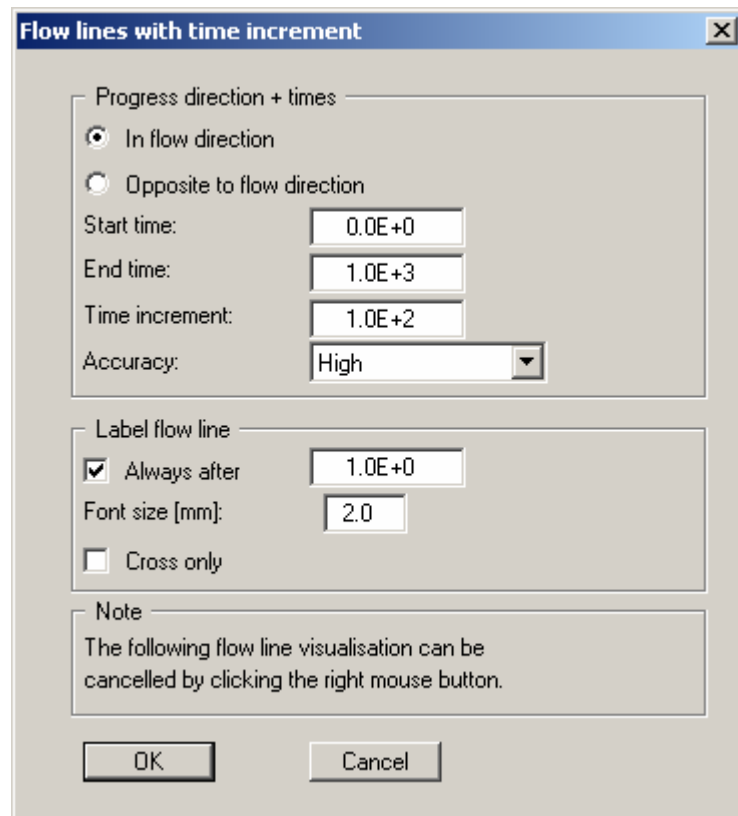
### 7.11.1 "Copyright" menu item

You will see a copyright message and information on the program version number.

The "**System**" button shows information on your computer configuration and the folders used by **GGU-3D-TRANSIENT**.

### 7.11.2 "Maxima" menu item

A box opens showing the maximum values allowed by the program for the nodes and elements of the FEM mesh.

### 7.11.3 "Help" menu item

The **GGU-3D-TRANSIENT** manual is opened as a PDF document. The help function can also be accessed using the [**F1**] function key.

### 7.11.4 "GGU on the web" menu item

Using this menu item you can access the GGU Software website: www.ggu-software.com.
Keep in touch with new program versions and the regular *download* offers.

If you would like to be automatically notified about program innovations, please register for the Newsletter in our Knowledge Base. Go to the following website: http://kbase.civilserve.com.

### 7.11.5 "GGU support" menu item

This menu item takes to the GGU-Software Support area at www.ggu-software.com.

### 7.11.6 "What's new?" menu item

You will see information on program improvements in comparison to older versions.

### 7.11.7 "Language preferences" menu item

This menu item allows you to switch the menus and the graphics from German to English and vice versa. To work in German, deactivate the two check boxes "**Dialoge + Menüs übersetzen (translate dialogs, menus)**" und "**Graphiktexte übersetzen (translate graphics)**".

Alternatively, you can work bilingually, e.g. with German dialog boxes but with graphic output in English. The program always starts with the language setting applicable when it was last ended.

# 8 Tips and tricks

## 8.1 Keyboard and mouse

You can scroll the screen with the keyboard using the cursor keys and the [**Page up**] and [**Page down**] keys. By clicking and pulling with the mouse, with [**Ctrl**] pressed, you activate the zoom function, i.e. the selected section will fill the screen. Use the mouse wheel to zoom in or out of the screen view or to pan.

In addition, scale and coordinates of the system graphics (drawing area within the plotting margins) can be altered directly using the mouse wheel. The following mouse wheel functions are available

**Change system graphics (new values can be checked in "Page size + margins/Manual resize (editor)"):**

- [**Ctrl**] + mouse wheel up = enlarge system graphics (change of scale)

- [**Ctrl**] + mouse wheel down = shrink system graphics (change of scale)

- [**Shift**] + mouse wheel up = move system graphics up
  (change in system coordinates)

- [**Shift**] + mouse wheel down = move system graphics down
  (change in system coordinates)

- [**Shift**] + [**Ctrl**] + mouse wheel up = move system graphics right
  (change in system coordinates)

- [**Shift**] + [**Ctrl**] + mouse wheel down = move system graphics left
  (change in system coordinates)

**Change screen coordinates:**

- Mouse wheel up = move screen image up

- Mouse wheel down = move screen image down

- [**Alt**] + [**Ctrl**] + mouse wheel up = enlarge screen image (zoom in)

- [**Alt**] + [**Ctrl**] + mouse wheel down = shrink screen image (zoom out)

- [**Alt**] + [**Shift**] + mouse wheel up = move screen image right

- [**Alt**] + [**Shift**] + mouse wheel down = move screen image left

If you click the right mouse button anywhere on the screen a context menu containing the principal menu items opens.

| |
|---|
| Normal contours |
| Coloured contours |
| Velocities |
| Discharges |
| Define section |
| Individual values |
| Potential external model |
| Seepage surface |
| Print and export |
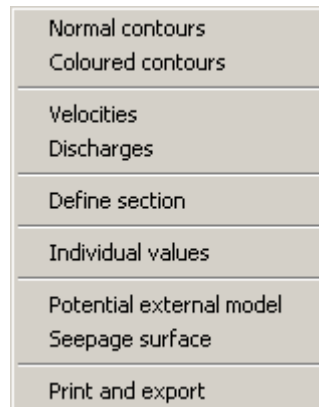
By double-clicking the left mouse buttonon legends or **Mini-CAD** objects, the editor for the selected object immediately opens, allowing it to be edited.

## 8.2    Function keys

Some of the function keys are assigned program functions. The allocations are noted after the corresponding menu items. The individual function key allocations are:

- [**Esc**] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.

- [**F1**] opens the manual file.

- [**F2**] refreshes the screen without altering the current magnification.

- [**F5**] opens the menu item "**System/Model**".

- [**F6**] opens the menu item "**Times/Read next time step**".

- [**F7**] opens the menu item "**Times/Read previous**".

- [**F9**] activates the menu item "**Page size + margins/Auto-resize**".

- [**F11**] activates the menu item "**Graphics preferences/Move objects**".

## 8.3 "Copy/print area" icon

A dialog box opens when the "**Copy/print area**" icon  in the menu toolbar is clicked, describing the options available for this function. For example, using this icon it is possible to either copy areas of the screen graphics and paste them into the report, or send them directly to a printer.

In the dialog box, first select where the copied area should be transferred to: "**Clipboard**", "**File**" or "**Printer**". The cursor is displayed as a cross after leaving the dialog box and, keeping the left mouse button pressed, the required area may be enclosed. If the marked area does not suit your requirements, abort the subsequent boxes and restart the function by clicking the icon again.

If "**Clipboard**" was selected, move to the MS Word document (for example) after marking the area and paste the copied graphics using "*Edit/Paste*".

If "**File**" was selected, the following dialog box opens once the area has been defined:
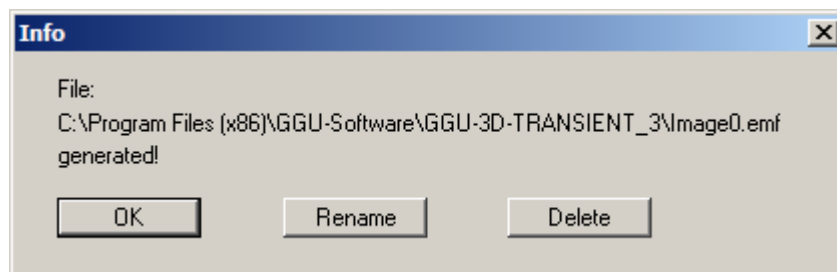


The default location of the file is the folder from which the program is started and, if several files are created, the file is given the file name "**Image0.emf**" with sequential numbering. If the "**Rename**" button in the dialog box is clicked, a file selector box opens and the copied area can be saved under a different name in a user-defined folder. Saving can be aborted by pressing the "**Delete**" button.

If the "**Printer**" button was pressed in the first dialog box, a dialog box for defining the printer settings opens after marking the area. Following this, a dialog box for defining the image output settings opens. After confirming the settings the defined area is output to the selected printer.

# 9 Index