Modelling of transient groundwater flow

GGU-TRANSIENT

VERSION 6

Contents:

1 Preface

The **GGU-TRANSIENT** program allows modelling of transient groundwater flow in horizontal-plane, vertical-plane and axis-symmetrical groundwater systems using the finite element method.

Data input is in accordance with conventional WINDOWS operations and can therefore be learned almost entirely without the use of a manual. In order to avoid unnecessarily inflating the program, no evaluation routines have been included in **GGU-TRANSIENT**. For evaluation and presentation of transient groundwater flow, please see the **GGU-PLGW** program. Using **GGU-PLGW** you can, for instance, generate a coloured animation, reflecting the flow development in specified time intervals. Furthermore, **GGU-PLGW** allows presentation of the various flow quantities along sections through the modelled system, as well as the temporal changes in these values (hydrographs) at specified points in the system.

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-SS-FLOW2D** 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-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-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-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 "**?**" 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-TRANSIENT** acquires this initial condition from a record (e.g. "**Gwh1.da1**" or "**Gwv1.da1**") generated during a previous analysis using the **GGU-SS FLOW2D** 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.) <u>cannot</u> be altered from within the **GGU-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 as a function of the type of groundwater system (vertical-plane, horizontal-plane, etc.).

Further to these additional soil properties, time-dependent boundary conditions can also be defined with **GGU-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 as so-called boundary data, in order to make them available again for later modelling.

Boundary conditions defined in the course of steady-state modelling (**GGU-SS-FLOW2D**) are also contained in the file (e.g. "**Gwh1.da1**"). They are assumed by the program to be constant for the whole modelling duration, unless the so-called *fixed* boundary conditions are deleted in **GGU-TRANSIENT**.

The **GGU-TRANSIENT** program allows presentation of the steady-state file results (e.g. "**Gwh1.da1**") on the screen, facilitating quick and easy control. Transient data <u>cannot</u> be evaluated using the **GGU-TRANSIENT** program. The **GGU-PLGW** program is available for this purpose; it is described in a separate user manual.

Generally, when modelling transient groundwater systems, a flood of result values is generated. Modelling results are saved by the **GGU-TRANSIENT** program in a file "**_t.plw**". The file containing the base data (polygon courses, etc.) has the name "**_ t.dat** ". When modelling is complete, this file is automatically saved and then automatically reloaded when the **GGU-SS-FLOW2D** file ("**.da1**") is reloaded.

The **GGU-TRANSIENT** program is started using conventional WINDOWS operations. The program is equipped with a large number 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.

5 Starting the program

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

- File

- ?

After clicking on the "**File**" menu, a system processed with the **GGU-SS-FLOW2D** program can be loaded by means of the "**Load GW data**" menu item.

> The processed system record is saved in **GGU-SS FLOW2D** using the "**.da1**" extension, in order to make it available for **GGU-TRANSIENT**. The program function to be activated for this purpose can be found in **GGU-SS-FLOW2D** in the "**System/Analyse**" menu item.

Then, the menu bar will show eight menus:

- File

- Time BC

- Fixed BC

- Check

- Modelling

- Graphics preferences

- Page size + margins

- ?

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-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.

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-SS-FLOW2D** groundwater flow program for transient modelling using the **GGU-TRANSIENT** program, you must create a file in the **GGU-SS-FLOW2D** program with the file extension "**.da1**". You can create this file by going to "**System/Analyse**" and activating the check box "**Generate record for transient modelling**".

- Start the **GGU-TRANSIENT** program and, using the menu item "**File/Load GW data**", select the system from which you would like to analyse 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-SS-FLOW2D**. This list is supplemented by information on the specific storage coefficient. In the case of a vertical-plane system, you can define your specific soil curves, using modelling points and according to soil type, by clicking on the "**nw = f(u)**" field (*cf.* Figure 1 in Section 7.2.3).

- Using the menu item "**Time BC/Edit polygon courses**" 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 "**Change 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 "**Modelling/Start**" menu item the program analyse the transient groundwater flow with the user-defined boundary conditions. You can specify the final time and the temporal increment in the dialog box.

- Use the **GGU-PLGW** program for evaluation and visualisation of the modelling results; it also provides a means of printing an impressive graphic of the results. To do this, load the file generated by **GGU-TRANSIENT** ("**_t.plw**") into **GGU-PLGW**. An initial evaluation can also be carried out parallel to modelling.

7 Description of menu items

7.1 File menu

7.1.1 "Load GW data" menu item

Existing steady-state groundwater modelling data generated with the **GGU-SS-FLOW2D** program can be loaded into **GGU-TRANSIENT**. This file has the default name of "**.da1**". In order to enable generation of this record by the **GGU-SS-FLOW2D** program, you must activate the "**Generate record for transient modelling**" check box in the "**System/Analyse**" menu item. If you are not sure which system is in the "**.da1**" file, go to the "**Check**" menu.

7.1.2 "Save as" menu item

For parameter studies - e.g. when adopting various polygon courses - it is advisable to generate individual "**.da1**" initial records, because you cannot influence the naming of your transient analysis result files. The program stores the base data automatically in a "**.dat**" file under the name of the initial file and extended by a "**_t**". The results required for subsequent evaluation in **GGU-PLGW** are automatically saved in a "**_t.plw**" file. It is therefore possible that different transient systems are mistakenly exported to the same result files if the same initial record is used.

Using this menu item the data are saved in an existing or a new file with the file extension "**.da1**". If you do not enter an extension when saving "**.da1**" will be used automatically.

7.1.3 "System info" menu item

Upon opening this menu item you will be presented with information on the currently loaded system, the number of triangular elements and on the nodes.

7.1.4 "Load boundary values" menu item

Using this menu item you can reload a "**.dat**" 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 modelling. 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 "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.

7.1.7 "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.
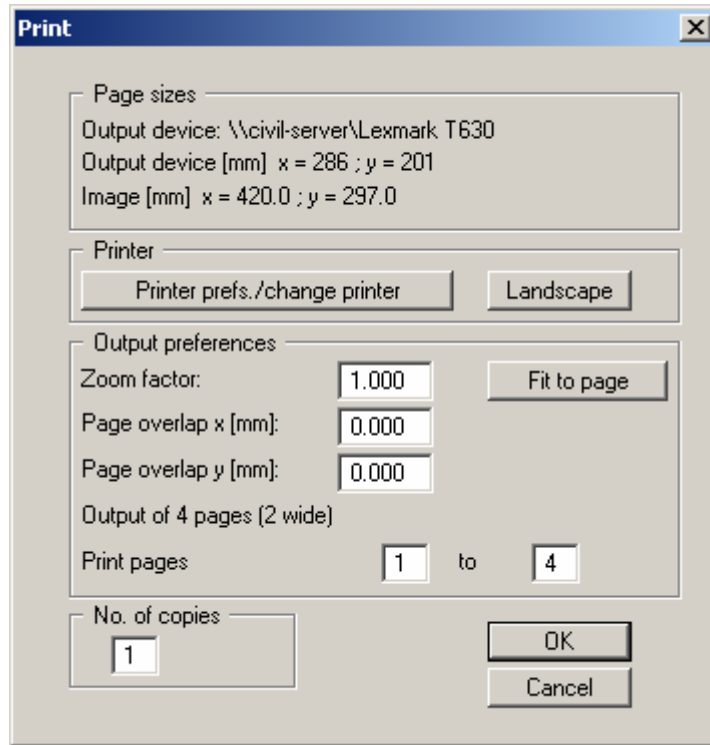
In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate 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.

7.1.8   "Exit" menu item

After a confirmation prompt, you can quit the program.

7.1.9   "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   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-SS-FLOW2D**.

7.2.2   "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.

7.2.3   "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-SS-FLOW2D** program can no longer be amended here

Horizontal-plane systems:

For a horizontal-plane system the dialog box displays the soil properties defined by the user in **GGU-SS-FLOW2D**. It includes a further variable requiring user input: the specific storage coefficient of the fluid and the grain skeleton Ss.
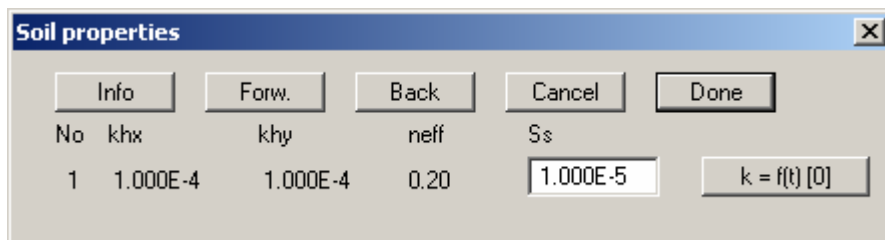
The soil permeability can be altered as a function of the time using the "**k = f(t) [0]**" button.

Vertical-plane and axis-symmetrical systems:

The dialog box for a vertical-plane or axis-symmetrical system corresponds to the previous box. Here, too, the steady-state soil properties are supplemented by the specific storage coefficient of the fluid and the grain skeleton Ss.
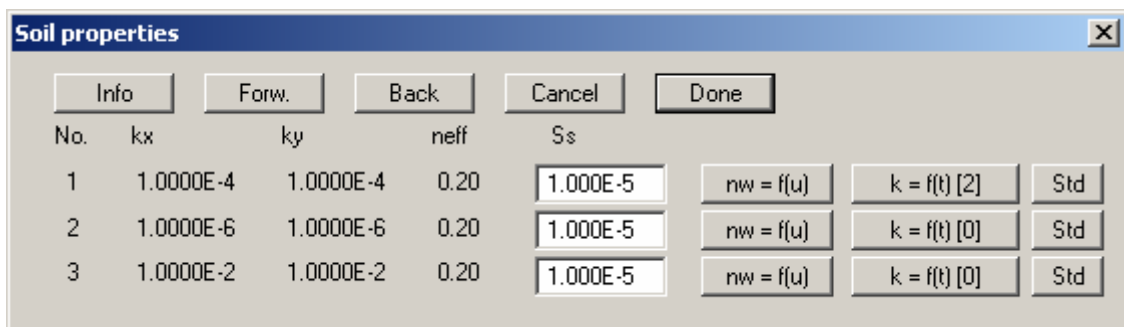
The value for n_w can also be entered here. The value of n_w gives the volume of water relative to the total volume. It is a dimensionless variable and is a function of the pore water pressure u. Typical function curves for various soil types are shown in Figure 1:
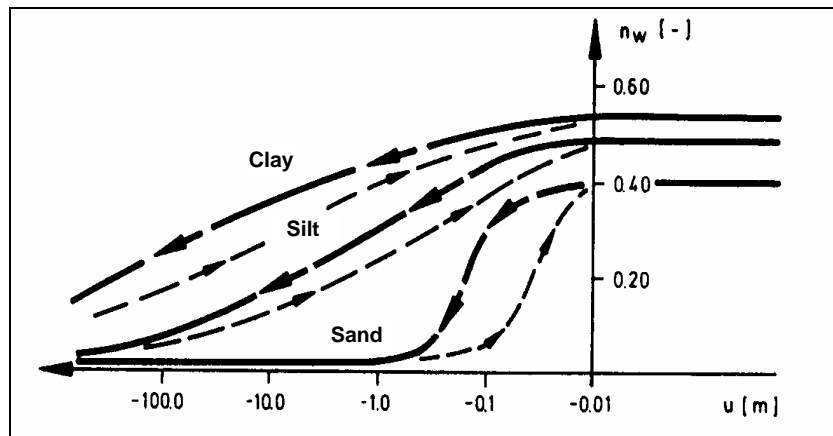
Figure 1 Function curves nw = f(u)

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 nw 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 "**Apply to 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.

7.2.4 "Edit polygon courses" menu item

The temporally variable (transient) system boundary conditions for the system are defined using this menu item. Previously defined polygon courses may also be edited. Polygon course input is independent of location. Existing polygon courses are opened in this menu item's dialog box by pressing the button with the corresponding polygon course number. The same dialog box opens as for the "**New**" button, 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.

- "**Change polygon course**"
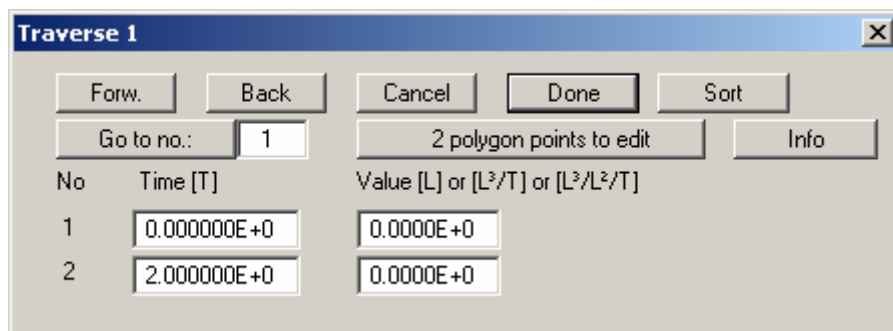
You define the number of polygon course modelling points via the "**2 polygon points to edit**" button. It is possible to define up to 100 polygon points. Once you have done this you can define the temporal course with your corresponding boundary condition value. Take care to note the units you are using. 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.

- "**Display 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.
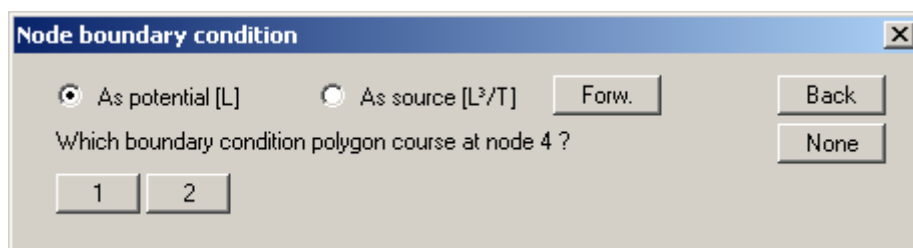

- "**Load polygon course**"
  The data from a previously saved file can be loaded (e.g. from a previous project or a different system).

- "**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).

- "**Polygon course via sinus function**"
  This allows you to generate a polygon course via a sinus 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.

7.2.5 "Delete all" menu item

You can delete all polygon courses.

7.2.6 "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**". 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.

7.2.7 "(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 "**Define**" button is pressed the same dialog box opens as for "**Time BC/Point potential or source**" (see Section 7.2.6).

7.2.8 "Linear (potential or source)" 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 helpful for defining a receiving watercourse (horizontal-plane system) if its water level is linearly variable within the system.

7.2.9 "Linear sources" menu item

The procedure for defining time-dependent linear sources is explained in a dialog box. After clicking system nodes and confirming the selection using the [**Return**] key, a dialog opens for selecting a polygon course.
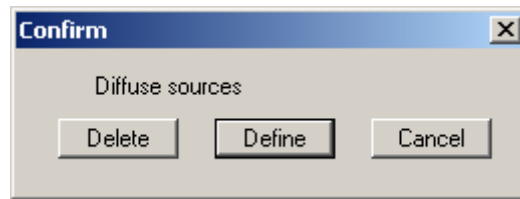
If linear sources have already been defined an additional "**Delete all linear sources**" button appears in the info box. If the dialog box is closed using this button, all linear sources are deleted and a new linear source can be defined. If individual linear sources or sections of them need to be deleted, click the system node once again and exit the polygon course allocation dialog box by pressing "**OK**". The following dialog box now appears:

Confirm	✕
Section already has a linear source !	
Delete old source	Cancel

Click the "**Delete old source**" button until all sections of the marked linear source are deleted and then, if required, define the new linear source. If you exit the box by pressing "**Cancel**" before you are finished, the undeleted boundaries will remain in place.


7.2.10 "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 "**Define**" 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**".

7.2.11 "Diffuse sources for soils" menu item

You can assign time-dependent diffuse sources to certain soil types. Select the required soil in the dialog box and click the "**Assign**" button.
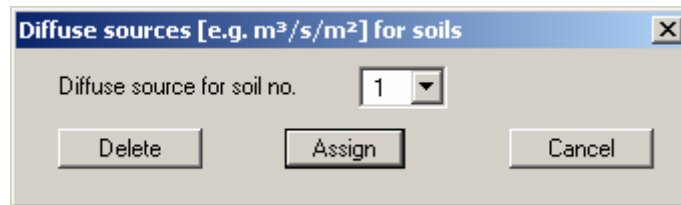
In a further dialog box a user-defined boundary condition polygon is selected to which this soil no. is to be assigned. Existing diffuse sources for certain soils can be removed from the system once again using the "**Delete**" button shown in the above dialog box.

7.2.12 "Check" menu item

The FEM mesh is displayed on the screen. Nodes and elements assigned a boundary condition are marked with the polygon course no. and a letter corresponding to the type of boundary condition. If you now click a marked node or an element with the left mouse button the polygon course will be displayed.

If you wish to view several nodes or elements the function can be called very quickly using the [**F3**] key. In this way you can jump directly from the hydrograph visualisation to the mesh visualisation and then click any other node.

7.2.13 "Display all polygon courses" menu item

You can have all user-generated polygon courses displayed as hydrographs in a single image.

7.2.14 "Special 1" menu item (for vertical-plane or axis-symmetrical systems only)

This menu item is only available for vertical-plane and axis-symmetrical systems. You will see the following information:
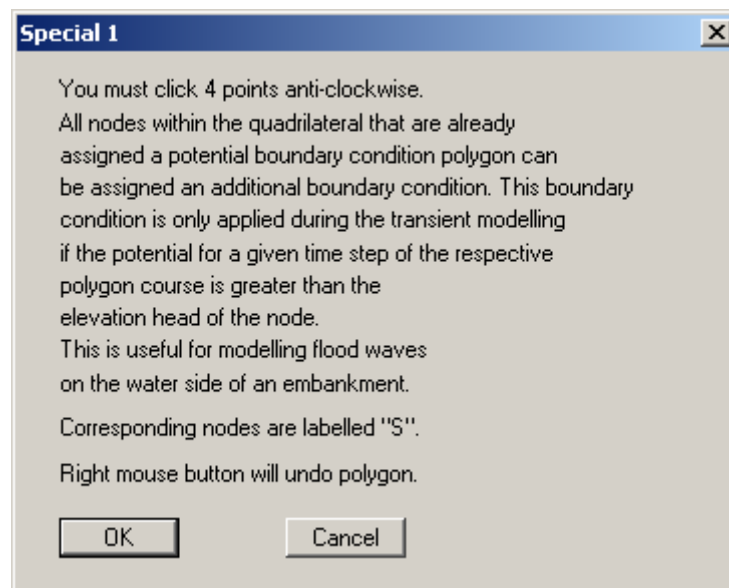

You can thus restrict the occurrence of boundary conditions This can be useful in order to avoid a *negative pressure* in the system.

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.
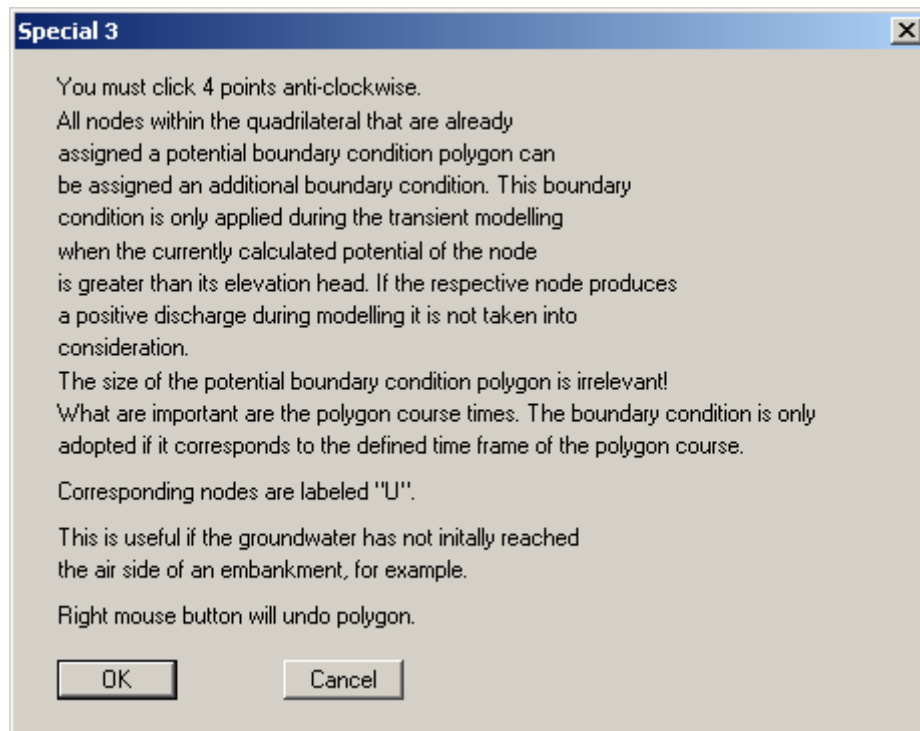
7.2.15 "Special 2" menu item (for vertical-plane or axis-symmetrical systems only)

The info box explains the functions.
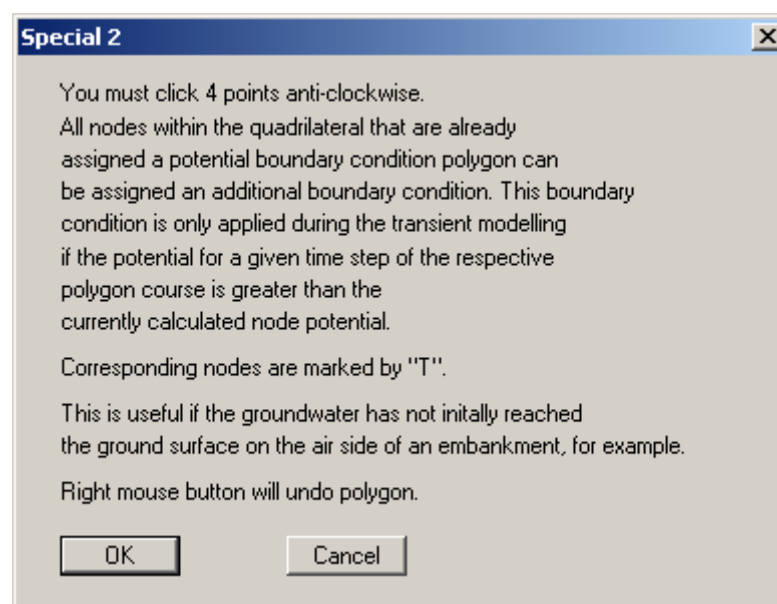
7.2.16 "Special 3" menu item (for vertical-plane or axis-symmetrical systems only)

A further info box explains the functions in this menu item.

Special 3

You must click 4 points anti-clockwise.
All nodes within the quadrilateral that are already
assigned a potential boundary condition polygon can
be assigned an additional boundary condition. This boundary
condition is only applied during the transient modelling
when the currently calculated potential of the node
is greater than its elevation head. If the respective node produces
a positive discharge during modelling it is not taken into
consideration.
The size of the potential boundary condition polygon is irrelevant!
What are important are the polygon course times. The boundary condition is only
adopted if it corresponds to the defined time frame of the polygon course.

Corresponding nodes are labeled "U".

This is useful if the groundwater has not initally reached
the air side of an embankment, for example.

Right mouse button will undo polygon.

[OK]   [Cancel]

7.2.17 "Blanket layer water level in section" menu item (for leaky aquifer only)

Certain elements can be assigned time-dependent water levels for the blanket layer by defining the area required as a polygon using 4 anti-clockwise points. You will then see the following dialog box:
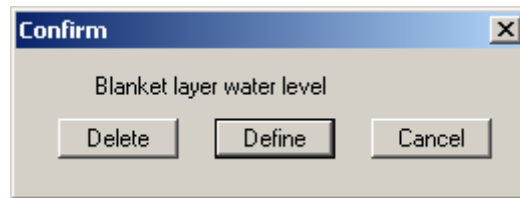
If water levels have already been defined they can be deleted from the marked elements by pressing the "**Delete**" button. After clicking the "**Define**" button a dialog box opens for assigning a polygon course.

7.2.18 "Linear (blanket layer water level)" menu item (for leaky aquifer only)

After going to this menu item click the nodes of the FEM mesh to which time-dependent, linearly variable water levels 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.

7.2.19 "Show (blanket layer water level)" menu item (for leaky aquifer only)

The FEM mesh is displayed on the screen. FEM elements assigned a water level in the blanket layer as a boundary conditions are marked with the polygon course no. and a "**D**". If a marked field is now clicked using the left mouse button the polygon course will be displayed.

If you wish to view several elements the function can be called very quickly using the [**F4**] key. In this way you can jump directly from the hydrograph visualisation to the mesh visualisation and then click any other FEM element.

7.3 Fixed BC menu

7.3.1 General notes on "fixed" boundary conditions

The **GGU-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.

7.3.2 "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).

7.3.3 "Check" menu item

Here you can check the type of temporally fixed boundary condition defined at each node.

7.3.4 "Point potential or source" 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 or sources, similarly to the time boundary conditions, which remain constant for the analysis duration. In the subsequent dialog box you define the value and type of boundary condition. However, please observe the remark above (see Section 7.3.1).

7.3.5 "In section" menu item

Using this menu item you can define or delete potentials or sources in section. As described in the dialog box, click on a section with four points in an anti-clockwise direction using the left mouse button and then enter the type and value of the boundary condition. The definition then applies to the nodes within the section.

7.3.6 "Linear sources" menu item

The procedure for defining linear sources is explained in a dialog box. Further information can be found in the menu item "**Time BC/Linear sources**" (see Section 7.2.9).

7.3.7 "Diffuse sources" menu item

Certain elements can be assigned *fixed* diffuse sources by defining the area required as a polygon using 4 counter-clockwise points. You will then see a dialog box in which existing diffuse sources can be deleted from the marked elements by pressing the "**Delete**" button. After clicking the "**Define**" button a dialog box opens for entering the value for the diffuse source.

7.3.8 "Diffuse sources for soils" menu item

Individual soils can be defined as *fixed* diffuse sources by assigning a discharge to the respective soil (e.g. m³/s/m²). Select the soil in the dialog box and enter the value for the diffuse source.
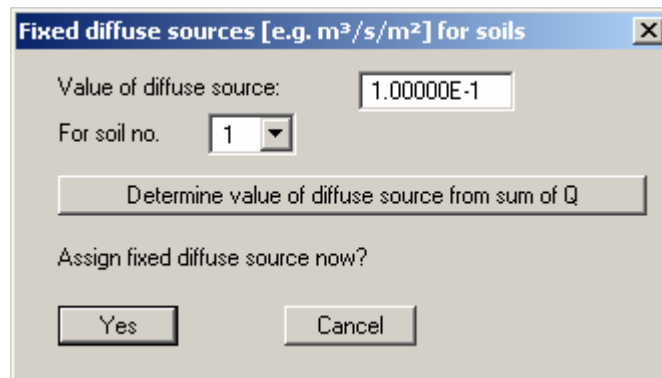
Alternatively, the value of the diffuse source can be automatically determined by the program. If you click the button "**Determine value of diffuse source from sum Q**" you must enter the discharge for the area of the selected soil in the new dialog box. The program then determines the value for the diffuse source from the data given. This value can then be accepted.

7.3.9 "Blanket layer water level" menu item (for leaky aquifer only)

Certain elements can be assigned *fixed* water levels for the blanket layer by defining the area required as a polygon using 4 anti-clockwise points. You will then see a dialog box in which existing water levels can be deleted from the marked elements by pressing the "**Delete**" button. After clicking the "**Define**" button a dialog box opens for entering the water level to be used.

7.4 Check menu

7.4.1 General note

The menu items in this menu serve to check the transient data and to aid your memory. You can have the mesh presented on the screen together with the desired parameters. With the following menu items you cannot edit any of the mesh analysis parameters.

7.4.2 "Preferences" menu item

In this dialog box you can specify preferences for the FEM mesh presentation.

7.4.3 "Mesh" menu item

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

7.4.4 "Outline" menu item

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

7.4.5 "Outline (coloured)" menu item

After going to this menu item the system is outlined and the soil colours displayed.

7.4.6 "Contours" menu item

After going to this menu item the contours of the potentials for the initial steady-state are displayed. A dialog box opens, in which the type of visualisation can be specified.

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.

The "**Further preferences**" button can be used to specify the type of visualisation of the contour lines and labelling.

7.4.7 "Contours (coloured)" menu item

After going to this menu item the contours of the potentials for the initial steady-state are displayed in colour. The following preferences can be defined in the dialog:

- "**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 7.7.5).

7.5.1 "Start" 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:

All results from the transient modelling are included in the output file. It is automatically given the "**_t.plw**" extension, because the **GGU-PLGW** evaluation program only displays files with this extension in the file selector box, for reasons of clarity.

Furthermore, 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 required increase time step via 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.

  During modelling, the program automatically proceeds to all times associated with boundary polygons, regardless of the selected time step. 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 output 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.

- Because long paths are shown shortened in the dialog box, the "**Display entire file name**" button can be clicked before commencing modelling. The name of the file containing the transient modelling data will then be displayed showing the complete path.

Once all data has been entered, start modelling by clicking "**OK**". During modelling, a dialog box appears with information on 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 data such as maximum deviation, time step, etc. and then carry on modelling.

A record with the extension "**_t.plw**" is generated for the evaluation application during modelling. If, after commencing modelling, you open the **GGU-PLGW** program, you can check and evaluate the current results while modelling is still in progress. The "**Animation**" function is particularly impressive here; it can be executed parallel to the running modelling process.

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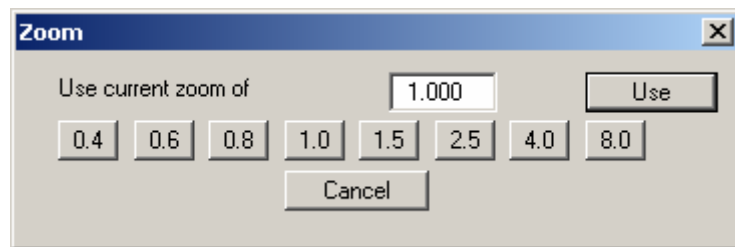
A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "**Use**" to exit the box the current factor is accepted. By clicking on the "**0.4**", "**0.6**", etc. buttons, the selected factor is used directly and the dialog box closed.

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

Using these two menu items you can add free text to the graphics and add lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.). 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.

7.6.6 "Toolbar preferences" menu item

After starting the program a horizontal toolbar for 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 smart icons 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-TRANSIENT.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the next time the program is started.

By clicking on the tools (smart icons) for the menu items you can directly reach most of the program functions. The meaning of the smart icons appears as a text box if you hover with the mouse pointer over the tools. Some of the tool functions cannot be activated from the normal menu items.

 "**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).

7.6.7 "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.
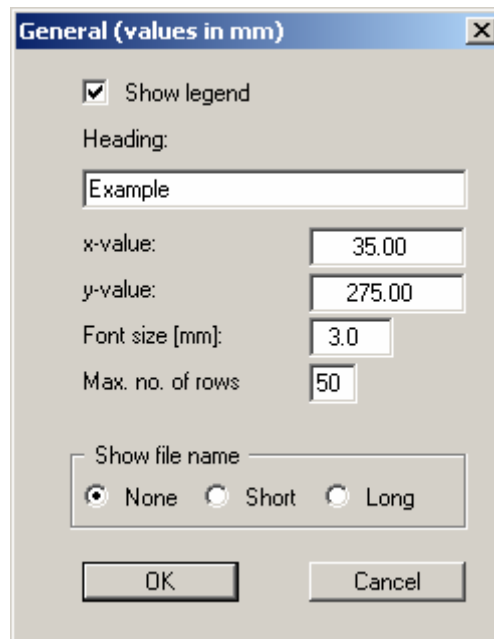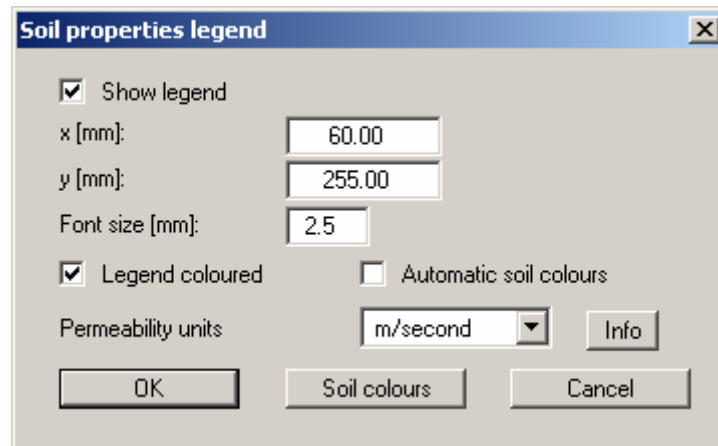

You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of rows**"; where necessary, several columns are used. In the general legend you can, if wished, display information on the file.

> 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.

7.6.8 "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.

- "**Permeability units**"
  It often makes more sense in a transient analysis to enter the permeability in "**m/hour**" or "**m/day**". If the unit used is selected in the dialog box above, the permeability entered is converted to "**m/second**" for use in the legend <u>only</u> (see "**Info**" button).

7.6.9 "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-SS-FLOW2D** 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.

7.6.10 "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.

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.

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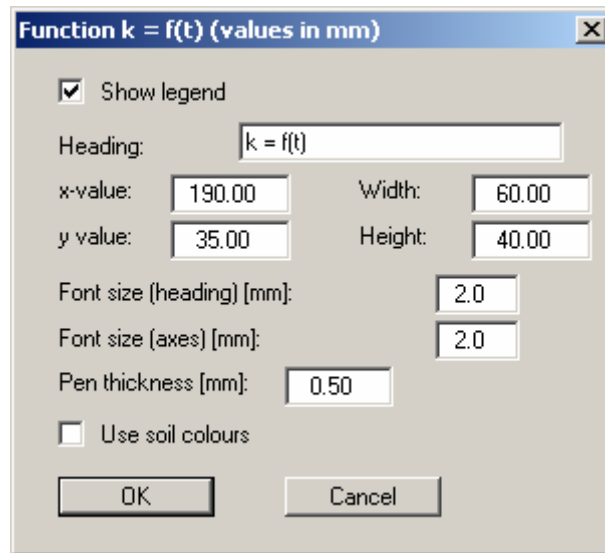
The position of the legend can be defined or edited by means of the variables "*x*" and "*y*", "**Width**" and "**Height**". If you edit the font sizes you should also always increase the width and height correspondingly, otherwise the actual function graphic will appear smaller within the defined legend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

If 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

7.6.12 "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.

7.6.13 "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-TRANSIENT.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**".

7.6.14 "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 7.7.5).

7.7.4 "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.

7.7.5 "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 size and margins                                    ×
  Edit page size and margins
   Page in general
   Height =      297.00        Width =      420.00

   Page margin in mm
   Left =        5.00          Right =      5.00
   Top =         5.00          Bottom =     5.00

   Plot margin in mm
   Left =        20.00         Right =      5.00
   Top =         50.00         Bottom =     15.00

        OK                          Cancel
```
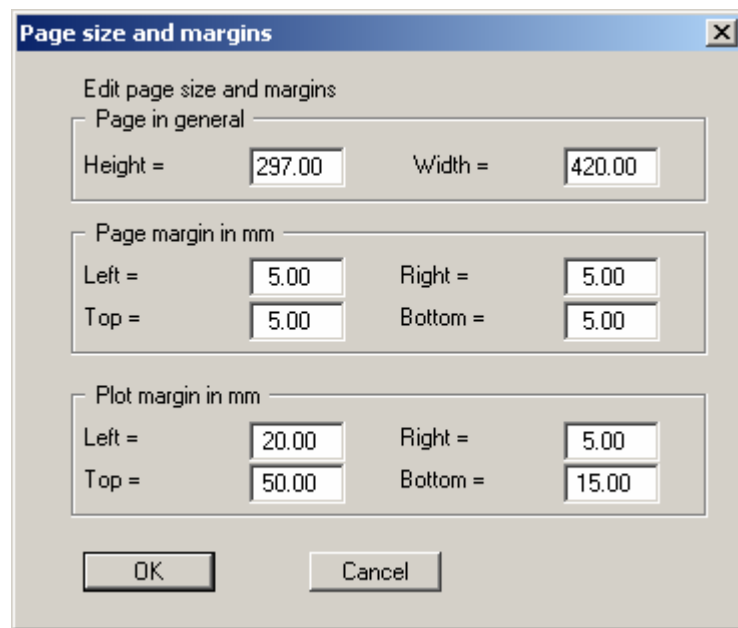
- "**Page in general**" defines the size of the output sheet. The A3 format is set as default. The program automatically draws thin cutting borders around the page, which are required when using a plotter on paper rolls.

- "**Page margin**" defines the position of a frame as a distance to the margins. This frame encloses the subsequent diagram.

- 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.

7.8.1   "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-TRANSIENT**.

7.8.2   "Maxima" menu item

A message box displays the program's default maximum number of nodes and the elements for the FEM mesh, polygon courses and the times within the polygon courses.

7.8.3   "Help" menu item

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

7.8.4   "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.

7.8.5   "GGU support" menu item

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

7.8.6   "What's new?" menu item

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

7.8.7   "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.

Soil properties
Edit polygon courses
Define point potential or source
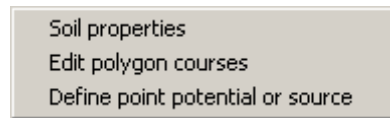
By double-clicking the left mouse button on legends or **Mini-CAD** objects, the editor for the selected element immediately opens, allowing it to be edited.

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.

- [**F3**] opens the menu item "**Time BC/Check**".

- [**F4**] opens the menu item "**Time BC/Show (blanket layer water level)**".

- [**F5**] opens the menu item "**Modelling/Start**".

- [**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:
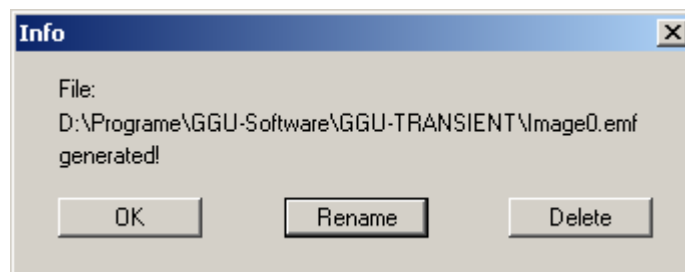
The default location of the file is the folder from which the program is started and, if several files are created, the file is given the file name "**Image0.emf**" with sequential numbering. If the "**Rename**" button in the dialog box is clicked, a file selector box opens and the copied area can be 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