

Graphical presentation and evaluation of  
groundwater flow

---

# GGU-PLGW

VERSION 7



---

Last revision: September 2008  
Copyright: GGU Zentrale Verwaltung mbH, Braunschweig  
Technical implementation and sales: Civilserve GmbH, Steinfeld

---

## Contents

<b>1 Preface .....</b>	<b>5</b>
<b>2 Licence protection and installation.....</b>	<b>5</b>
<b>3 Language selection.....</b>	<b>5</b>
<b>4 Starting the program .....</b>	<b>6</b>
<b>5 Short description.....</b>	<b>7</b>
<b>6 Description of menu items.....</b>	<b>8</b>
6.1 File menu.....	8
6.1.1 "Load" menu item .....	8
6.1.2 "Save" menu item .....	8
6.1.3 "Save as" menu item .....	8
6.1.4 "System info" menu item.....	8
6.1.5 "Export" menu item .....	8
6.1.6 "Differential data" menu item.....	8
6.1.7 "Printer preferences" menu item.....	9
6.1.8 "Print and export" menu item.....	9
6.1.9 "Batch print" menu item .....	11
6.1.10 "Exit" menu item .....	11
6.1.11 "1, 2, 3, 4" menu items.....	11
6.2 Evaluation menu.....	12
6.2.1 "Project identification" menu item.....	12
6.2.2 "Mesh" menu item .....	12
6.2.3 "Outline" menu item .....	12
6.2.4 "Potentials" menu item .....	12
6.2.5 "Potentials coloured" menu item.....	13
6.2.6 "3D potentials" menu item.....	14
6.2.7 "3D potentials (array)" menu item .....	16
6.2.8 "Velocities + gradients" menu item .....	17
6.2.9 "Potentials + velocities" menu item.....	18
6.2.10 "Discharges" menu item .....	18
6.2.11 "Discharge between 2 times" menu item .....	19
6.2.12 "Volume" menu item (only for "leaky aquifer").....	19
6.2.13 "Confined areas" menu item (only for "leaky aquifer").....	19
6.2.14 "GW thickness" menu item (only for "leaky aquifer") .....	19
6.2.15 "GW-surface distance" menu item (only for "leaky aquifer") .....	19
6.2.16 "Layer boundary contours" menu item (only for "leaky aquifer").....	20
6.2.17 "Layer thickness contours" menu item (only for "leaky aquifer").....	20
6.3 Flow lines menu .....	21
6.3.1 Principles .....	21
6.3.2 "Info" menu item .....	21
6.3.3 "Define individual points" menu item.....	21
6.3.4 "Define circle points" menu item.....	22
6.3.5 "Define end points" menu item.....	22

6.3.6	"Save points" menu item.....	22
6.3.7	"Load points" menu item .....	22
6.3.8	"Distance increment" menu item .....	22
6.3.9	"Time increment" menu item .....	22
6.4	Special menu .....	23
6.4.1	"Individual values" menu item.....	23
6.4.2	"Save ASCII file" menu item.....	23
6.4.3	"Define hydrograph points" menu item .....	23
6.4.4	"Display hydrographs" menu item .....	24
6.4.5	"Display position" menu item .....	24
6.4.6	"Save nodes" menu item .....	24
6.4.7	"Load nodes" menu item.....	24
6.4.8	"Define section" menu item .....	25
6.4.9	"Display section" menu item.....	25
6.4.10	"Display section course" menu item .....	25
6.4.11	"Save section" menu item .....	25
6.4.12	"Load section" menu item.....	25
6.4.13	"Phreatic lines" menu item.....	25
6.5	Graphics preferences menu .....	26
6.5.1	"Refresh and zoom" menu item .....	26
6.5.2	"Zoom info" menu item .....	26
6.5.3	"Pen colour and width" menu item .....	26
6.5.4	"Legend font selection" menu item.....	26
6.5.5	"Mini-CAD toolbar" and "Header toolbar" menu items .....	27
6.5.6	"Toolbar preferences" menu item .....	27
6.5.7	"3D toolbar" menu item .....	28
6.5.8	"General legend" menu item.....	28
6.5.9	"Soil properties legend" menu item .....	29
6.5.10	"Section legend" menu item.....	29
6.5.11	"kr = f(u) function legend" menu item.....	30
6.5.12	"nw = f(u) function legend" menu item .....	30
6.5.13	"k = f(t) function legend" menu item.....	31
6.5.14	"Hydrograph legend" menu item .....	31
6.5.15	"Hydrograph progress legend" menu item.....	32
6.5.16	"Move objects" menu item.....	32
6.5.17	"Save graphics preferences" menu item.....	32
6.5.18	"Load graphics preferences" menu item .....	32
6.6	Page size + margins menu .....	33
6.6.1	"Auto-resize" menu item.....	33
6.6.2	"Manual resize (mouse)" menu item.....	33
6.6.3	"Manual resize (editor)" menu item.....	33
6.6.4	"Page size and margins" menu item.....	34
6.6.5	"Font size selection" menu item.....	34
6.6.6	"Margins and borders" menu item .....	34

6.7	Times menu .....	35
6.7.1	"Read next time step" menu item.....	35
6.7.2	"Read previous" menu item .....	35
6.7.3	"Read specified" menu item.....	35
6.7.4	"Flip book" menu item.....	36
6.7.5	"Maximum values for all times" menu item .....	36
6.7.6	"Time format" menu item .....	37
6.7.7	"Month format" menu item .....	37
6.8	? menu .....	38
6.8.1	"Copyright" menu item.....	38
6.8.2	"Maxima" menu item.....	38
6.8.3	"GGU on the web" menu item .....	38
6.8.4	"GGU support" menu item.....	38
6.8.5	"What's new?" menu item.....	38
6.8.6	"Help" menu item .....	38
6.8.7	"Language preferences" menu item .....	38
<b>7</b>	<b>Tips.....</b>	<b>39</b>
<b>8</b>	<b>Index.....</b>	<b>41</b>

---

## 1 Preface

---

**GGU-PLGW** is a groundwater flow evaluation application for groundwater models created using the additional modules provided with the **GGU-SS-FLOW2D**, **GGU-TRANSIENT** (transient groundwater flow modelling) or **GGU-CONTAM-FE/GGU-CONTAM-RW** (contaminant transport modelling) applications. To keep things simple, only **GGU-TRANSIENT** will be named in the explanations laid out in this manual, even though the original files may have been created with one of the other programs.

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

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 note is important:

- The data forming the basis for evaluations are groundwater flow models created using the **GGU-SS-FLOW2D**, **GGU-TRANSIENT** and **GGU-CONTAM-FE/GGU-CONTAM-RW** applications.

---

## 2 Licence protection and installation

---

Because **GGU-PLGW** can only work in conjunction with records created by the above-named, licensed programs, **GGU-PLGW** itself does not require any further licensing.

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

---

## 3 Language selection

---

**GGU-PLGW** 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 Starting the program

---

If you have modelled groundwater flow using the **GGU-SS-FLOW2D** and **GGU-TRANSIENT** programs, you can evaluate these by starting the **GGU-PLGW** program 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.

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-TRANSIENT** program can be loaded by means of the "**Load**" menu item. Then, the menu bar will show eight menus:

- File
- Evaluation
- Flow lines
- Special
- Graphics preferences
- Page size + margins
- Times
- ?

After clicking one of these menus, the so-called menu items roll down, allowing you 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-PLGW** 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.

---

## 5 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 display transient groundwater flow. You can learn about the details of the program in the later chapters.

- In order to display transient groundwater flow with **GGU-PLGW**, a file with transient modelling results is required. This file is automatically generated, with the **"\*\_t.plw"** extension, when modelling transient flow with the **GGU-TRANSIENT** program.
- Open the **GGU-PLGW** program and select the required result file by means of the **"File/Load"** menu item.
- Go to the **"Evaluation"** menu. Here, you can specify which of your system flow quantities you would like to display and which type of presentation you prefer. The **"Potentials coloured"** menu item is especially impressive, or, for horizontal plane systems, the **"3D potentials (array)"** menu item. The subsequent dialogue boxes can almost always be left using **"OK"**, without further changes having to be made. The program will usually suggest sensible input. The **"Determine extreme values ..."** switch should be clicked once however, otherwise an error message will appear.
- Select the appropriate menu item for presentation of flow lines. Observe the information given in the preferences. You have the possibility of recording flow lines as distance or time increments.
- Using the menu item **"Special/Individual values"**, you can click on individual system nodes and have information on these presented.
- If your printer is a colour printer and is correctly installed in **WINDOWS**, you can create colour output by selecting the menu item **"File/Print and export"** and then selecting the **"Printer"** button in the subsequent dialogue box. For black and white printers, grey scales will be used. If necessary, alter the printer alignment in the dialogue box from **"Portrait"** to **"Landscape"** by clicking on the appropriate button. By clicking the **"Fit in"** button, a zoom factor is calculated, ensuring complete page output to the printer. Otherwise, several pages will be printed. After this, select the **"OK"** button in order to start printer output. You will then have a graphic output of your system in submission quality.
- Experiment with the example data files.

This short description shows that only a few items need to be selected for display of transient groundwater systems. All further menu items deal principally with saving data, and layout. More detailed descriptions follow in further chapters.

---

## 6 Description of menu items

---

### 6.1 File menu

---

#### 6.1.1 "Load" menu item

If, using the **GGU-TRANSIENT** program, you saved the output file of your transient system using the default "**\*\_t.plw**" extension, you can load the record using this menu item and evaluate, present or print the modelled groundwater flow.

#### 6.1.2 "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 without prompting with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

#### 6.1.3 "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 "**.plw**" 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, "**.plw**" will be used automatically.

#### 6.1.4 "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.

#### 6.1.5 "Export" menu item

The node coordinates can be saved to an ASCII file, allowing them to be imported into other programs where required.

If you are working with a vertical-plane system, an ASCII record can be exported especially for the GGU's **GGU-STABILITY** slope stability application. This contains the calculated potentials, beside the node coordinates. These data can be directly utilised in **GGU-STABILITY** as a pore-water pressure mesh.

#### 6.1.6 "Differential data" menu item

A differential record can be generated. The data in file 2, which is selected by pressing the "**GGU-PLGW2.plw**" button, is subtracted from the data in the current record (file 1). The differential record generated is saved in a file. The name of the file can be entered after pressing the "**Difference.plw**" button.

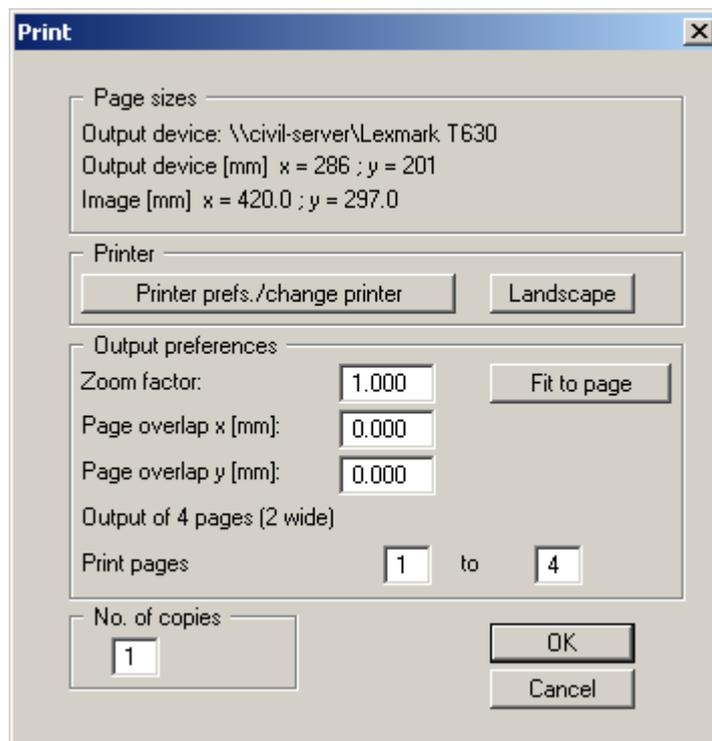
### 6.1.7 "Printer preferences" menu item

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

### 6.1.8 "Print and export" menu item

You can select your output format in a dialogue 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 dialogue 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.

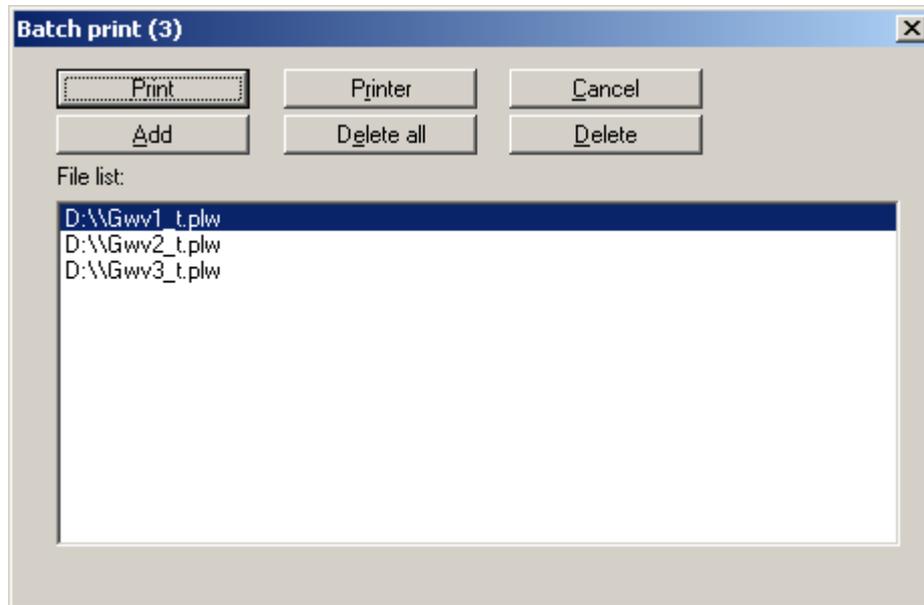
If you select the **"Copy/print area"** tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer.

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 MiniCAD module.
- **"GGUMiniCAD"**  
allows export of the graphics to a file in order to enable processing in the GGUMiniCAD program.
- **"Cancel"**  
Printing is cancelled.

### 6.1.9 "Batch print" menu item

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



Create a list of files for printing using "**Add**" and selecting the desired files. The number of files is displayed in the dialogue 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 dialogue 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.

### 6.1.10 "Exit" menu item

After a confirmation prompt, you can quit the program.

### 6.1.11 "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.

## 6.2 Evaluation menu

---

### 6.2.1 "Project identification" menu item

You can enter a description of the current system; this will then be used in the general legend (see Section 6.5.8).

### 6.2.2 "Mesh" menu item

Using this menu item you can define the appearance of the FEM mesh on the screen. The element no. and the soil no. cannot be displayed simultaneously.

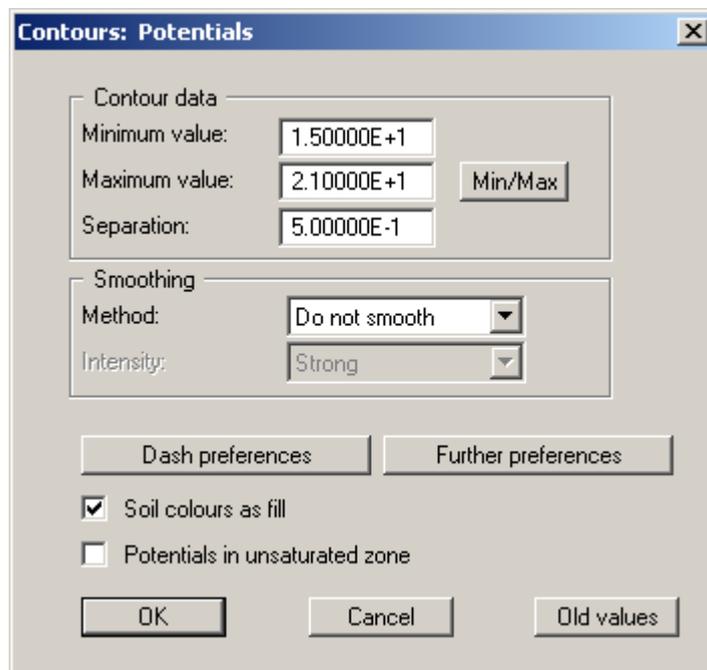
The "**Display mesh**" button produces a direct representation of the FEM mesh using the selected preferences.

### 6.2.3 "Outline" menu item

After going to this menu item the outlines of the various soils in the FEM system are displayed. If the coloured visualisation is activated under "**Evaluation/Mesh**", the outlines of the individual soils are filled with the defined colours.

### 6.2.4 "Potentials" menu item

You can visualise lines of equal water level (potentials) using this menu item. You will see the following dialogue 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:

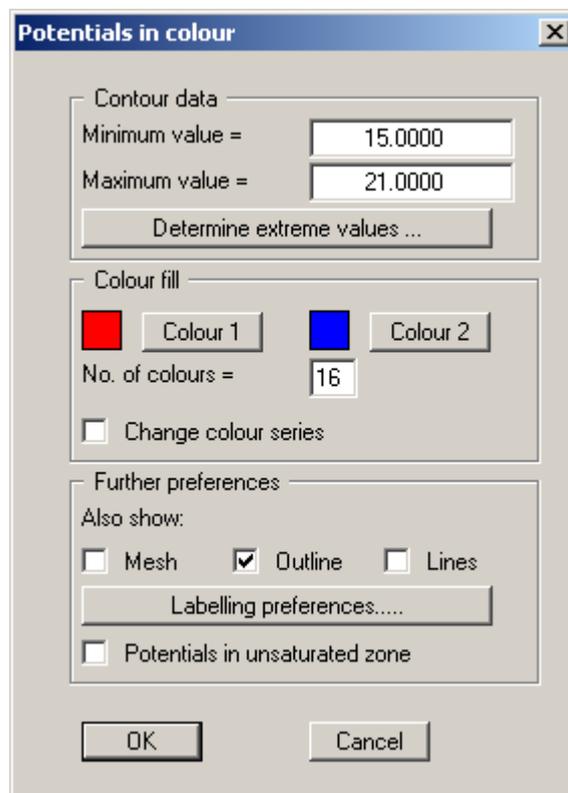
- "**Do not smooth**"  
As a linear method is implemented within the triangle elements, this will provide modelling results without smoothing performed by the program.
- "**Method 1**" and "**Method 2**"  
these smoothing out procedures employ two different Bezier splines. Smoothing out intensities can 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 "**Dash preferences**" and "**Further preferences**" buttons. By activating the appropriate check box the soil colour can be used as fill.

Because the program uses unsaturated permeabilities for its modelling of vertical-plane systems, potentials for the unsaturated zone are also acquired; physically, this is absolutely correct. Experience shows that this confuses some clients. In order to avoid lengthy discussions, simply switch off the potentials in the unsaturated zone using the appropriate check box.

### 6.2.5 "Potentials 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 dialogue box.

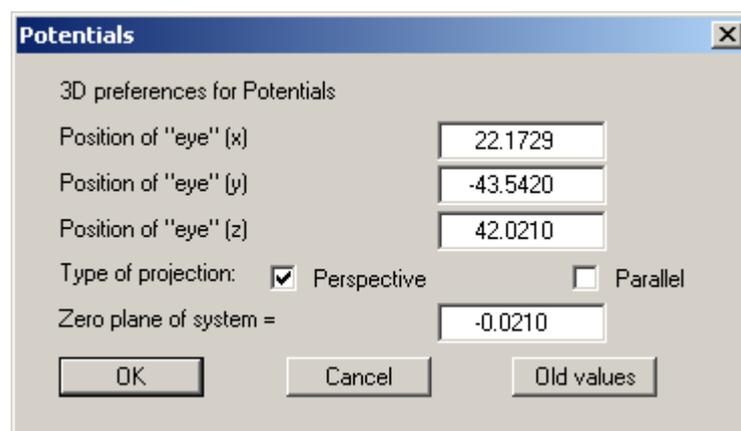
- "**Contour data**" group box  
Press the "**Determine extreme values...**" button. The program then determines the minimum and maximum values for the corresponding layer base. You can then edit these values, for example in order achieve a defined start value.
- "**Colour fill**" group box  
You can 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 6.6.4).

As the program calculates with unsaturated permeabilities for vertical plane systems, 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.

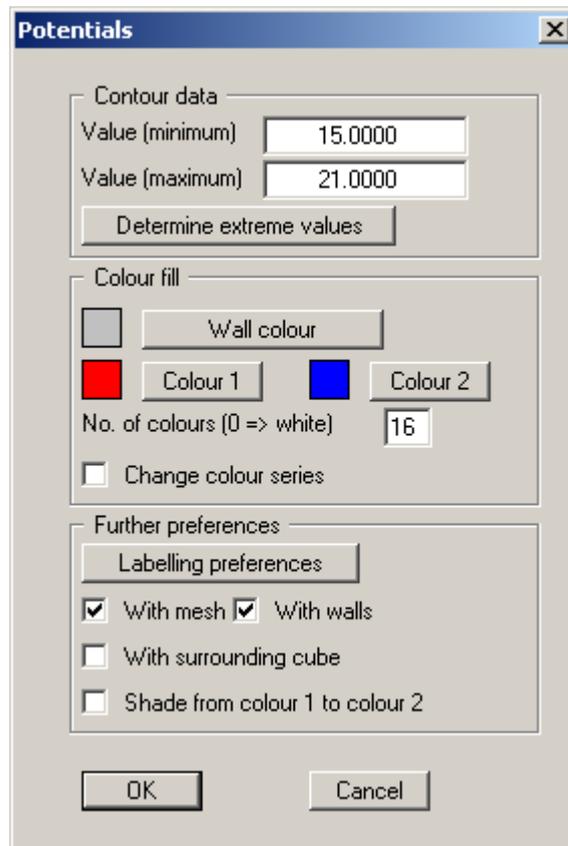
### 6.2.6 "3D potentials" menu item

Using this menu item it is possible to visualise lines of equal water level (potentials) for the existing nodes in three dimensions. In the following dialogue 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 dialogue box, so you will not normally need to make alterations.

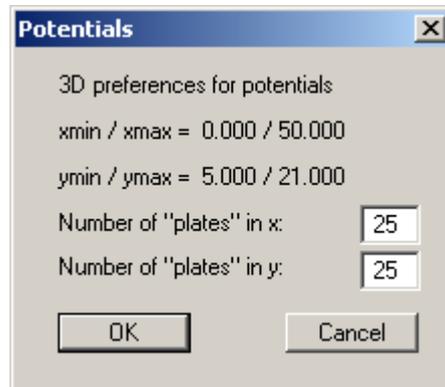
After leaving the dialogue box by pressing "OK" you will see the following dialogue box. It greatly resembles the box shown in the menu item "Evaluation/Potentials coloured".



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.

### 6.2.7 "3D potentials (array)" menu item

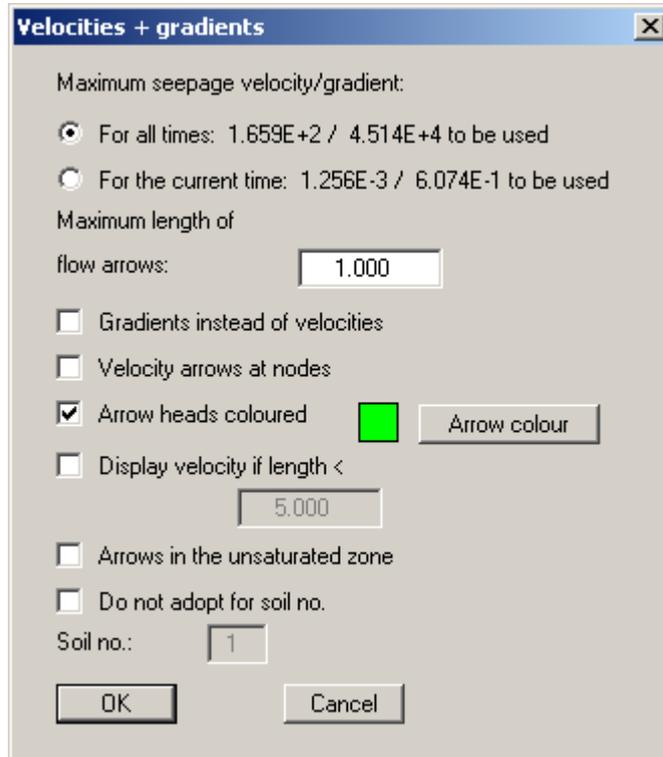
The 3D effect is usually lost in an irregular triangular mesh. The "**Evaluation/3D potentials (array)**" menu item is useful here. After selecting the required parameter the following dialogue box opens where you can define the array.



After leaving the dialogue box by pressing "**OK**" you will at first see the same box as that described in "**Evaluation/3D potentials**". The program calculates the state variables at the array points by linear interpolation of the results of the FEM modelling. You then see the dialogue box for coloured contours preferences as described in "**Evaluation/3D potentials**" (see Section 6.2.6).

## 6.2.8 "Velocities + gradients" menu item

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



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**" switch, 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 calculation. 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, at the top left, scale information for length/time units.

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.

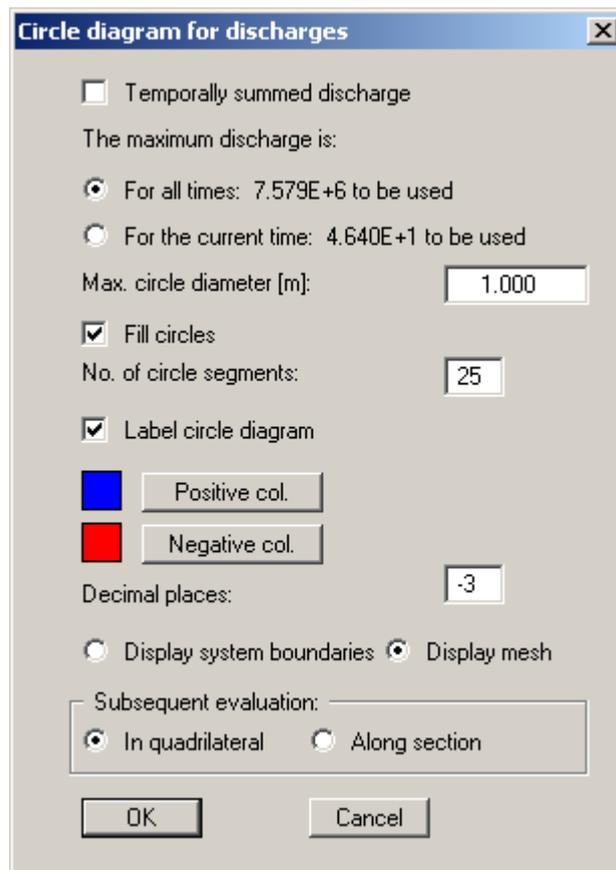
As the program calculates with unsaturated permeabilities for vertical plane systems, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone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

### 6.2.9 "Potentials + velocities" menu item

Potentials and flow velocities can also be displayed simultaneously, thus illustrating their interdependency. The dialogue boxes for contour line and velocity display preferences are shown in sequence. The dialogue box descriptions can be found in Sections 6.2.4 and 6.2.7.

### 6.2.10 "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 dialogue 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 are displayed in different colours in filled circles.

By specifying appropriate areas of the screen, the sum of the discharges in these areas can be calculated by the program. If the "**In quadrilateral**" check box is activated when leaving the dialogue box to continue the evaluation, the region for which the discharge is to be determined must be defined by clicking four points in an anti-clockwise direction. The calculated results are displayed in an info box.

A further option is to have the discharge along any section calculated by activating the "**Along section**" check box before continuing to the subsequent evaluation. An info box containing the results for the section is displayed.

If a horizontal-plane system with a *leaky aquifer* is being investigated, the discharge for the blanket layer only may be calculated by activating the additional "**Discharge through blanket**" check box.

#### 6.2.11 "Discharge between 2 times" menu item

This menu item allows you to determine the discharge within a given time period.

If a horizontal-plane system with a *leaky aquifer* is being investigated, the discharge for the blanket layer only may be calculated by activating the additional "**Discharge through blanket**" check box.

#### 6.2.12 "Volume" menu item (only for "leaky aquifer")

You see a message box containing information on the groundwater-filled volume of your system.

#### 6.2.13 "Confined areas" menu item (only for "leaky aquifer")

Nodes in areas of the system where confined groundwater conditions are prevalent are marked with the letter "G". Alternatively, these areas can be colour-coded.

#### 6.2.14 "GW thickness" menu item (only for "leaky aquifer")

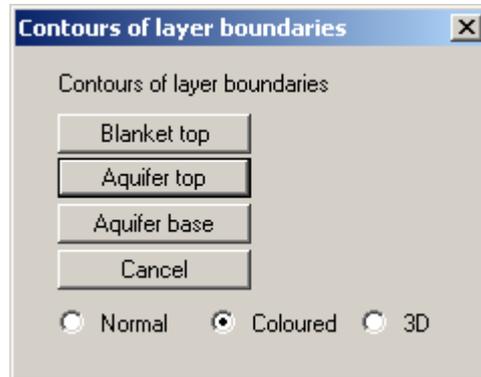
A normal or colour-filled contour diagram showing the groundwater thicknesses is generated (see menu items "**Evaluation/Potentials**", Section 6.2.4 and "**Evaluation/Potentials coloured**", Section 6.2.5).

#### 6.2.15 "GW-surface distance" menu item (only for "leaky aquifer")

A normal or colour-filled contour diagram showing lines of equal groundwater-surface distance is generated (see menu items "**Evaluation/Potentials**", Section 6.2.4 and "**Evaluation/Potentials coloured**", Section 6.2.5).

### 6.2.16 "Layer boundary contours" menu item (only for "leaky aquifer")

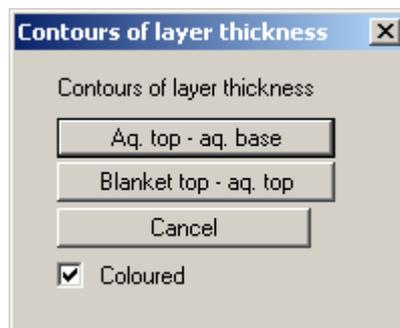
In order to clearly present the layer situation it is possible to generate a contour diagram of the top of the aquifer (aq. top) or the base of the aquifer (aq. base) or the top of the blanket layer (blanket top) for a *leaky aquifer*.



Define the type of contour diagram using the "**Normal**", "**Coloured**" and "**3D**" option buttons. Details are described in Sections 6.2.4 to 6.2.6. Exit the dialogue box by clicking the button for the layer for which the contour diagram is to be displayed.

### 6.2.17 "Layer thickness contours" menu item (only for "leaky aquifer")

You can create a contour diagram of the layer thicknesses.



If the "**Coloured**" switch is activated, colour filled contours are created.

## 6.3 Flow lines menu

---

### 6.3.1 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 starting point, is observed. The program determines size and direction of seepage from this starting 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 starting 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 ( $v_s = (k * i)/n_{eff}$ ). Using this program, you can also follow a particle back (against flow direction).

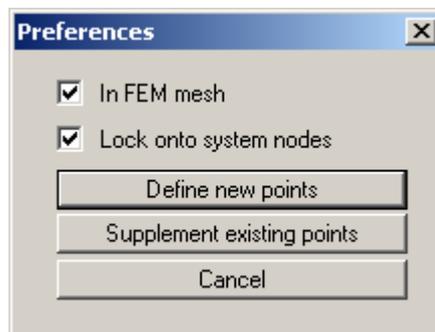
### 6.3.2 "Info" menu item

This menu item displays similar information on the screen to that explained above.

### 6.3.3 "Define individual points" menu item

You can define the starting points of water particles.

The following dialogue box appears for this menu item and the two following menu items, "**Flow lines/Define circle points**" and "**Flow lines/Define end points**".



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.

#### 6.3.4 "Define circle points" menu item

You can define several starting 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**")

Further descriptions concerning the dialogue box for this menu item can be found in Section 6.3.3 (menu item "**Flow lines/Define individual points**").

#### 6.3.5 "Define end points" menu item

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.

Further descriptions concerning the dialogue box for this menu item can be found in Section 6.3.3 (menu item "**Flow lines/Define individual points**").

#### 6.3.6 "Save points" menu item

You can save starting points to a file, in order to have them available for a different system.

#### 6.3.7 "Load points" menu item

Starting points saved using the previous menu item can be reloaded into the program using this button.

#### 6.3.8 "Distance increment" menu item

You can commence the modelling of a particle track (or tracks). The modelling parameters and the type of subsequent presentation can be confirmed or edited in the dialogue box. Activate the "**Against flow direction**" check box in order to retrace the path of a particle.

#### 6.3.9 "Time increment" menu item

You can commence the modelling of a particle track (or tracks). The modelling parameters and the type of subsequent presentation can be confirmed or edited in the dialogue box. Activate the "**Against flow direction**" check box in order to retrace the path of a particle.

## 6.4 *Special menu*

---

### 6.4.1 "Individual values" menu item

By clicking a FEM node using the left mouse button all modelling results for this node can be viewed.

### 6.4.2 "Save ASCII file" menu item

If, for example, you would like to create a table in your word processor using the individual values, export the data using this menu item to make them available to your word processor

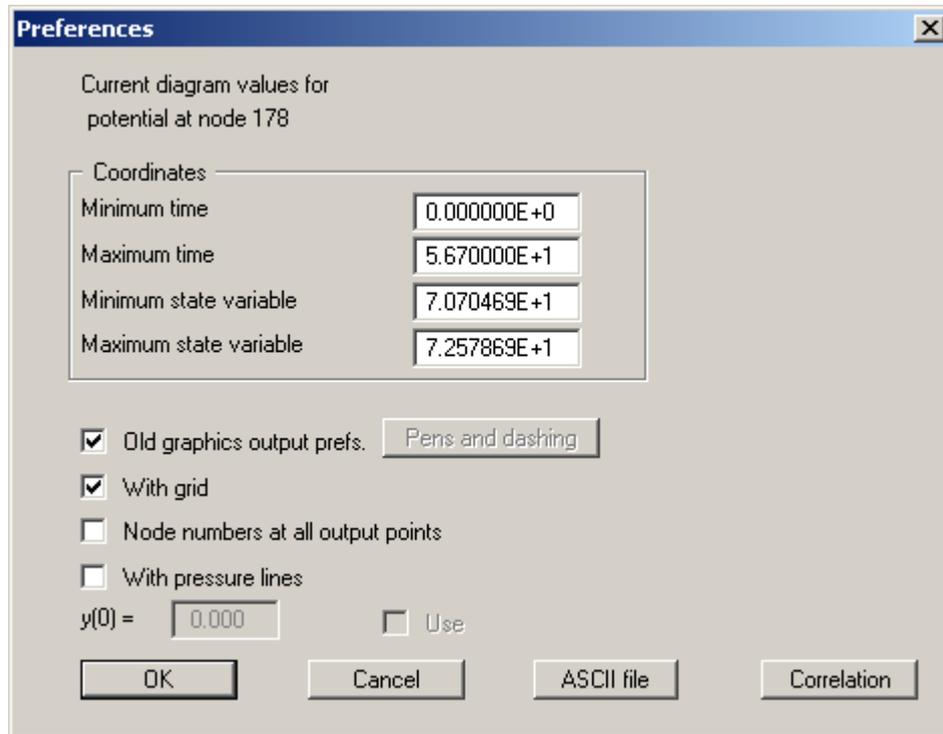
### 6.4.3 "Define hydrograph points" menu item

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 info 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 dialogue box. After clicking "**Define new nodes**" the existing nodes are deleted from the list.

#### 6.4.4 "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 dialogue 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.

#### 6.4.5 "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.

#### 6.4.6 "Save nodes" menu item

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

#### 6.4.7 "Load nodes" menu item

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

#### 6.4.8 "Define section" menu item

By clicking on system nodes with the left mouse button, you define the desired section course. The node points will then be numbered. Furthermore, the current mouse position in x and y coordinates is given in the top window bar. To delete the last node point specified, click on the point with the right mouse button.

#### 6.4.9 "Display section" menu item

A dialogue box appears with the option of selecting between displaying the current time or all times. You are then asked which variable to display in section. You may select between the potential, the pressure or the seepage velocity and the gradient in x and y directions.

#### 6.4.10 "Display section course" menu item

Using this menu item the section course can be displayed in your system on the screen.

#### 6.4.11 "Save section" menu item

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

#### 6.4.12 "Load section" menu item

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

#### 6.4.13 "Phreatic lines" menu item

The phreatic lines can be displayed for vertical-plane and axis-symmetrical systems. Specify the time step and the labelling format in the dialogue box.

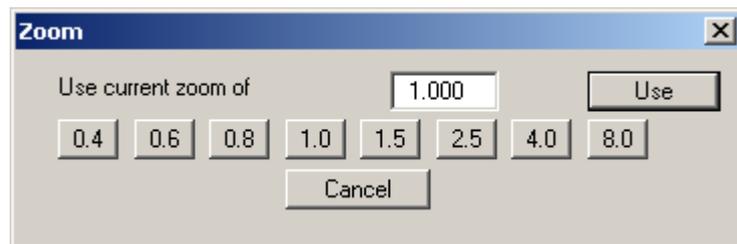
## 6.5 Graphics preferences menu

---

### 6.5.1 "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 dialogue 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.

### 6.5.2 "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.

### 6.5.3 "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, discharges, etc.). You can edit the pen widths for the elements shown in the dialogue 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.

### 6.5.4 "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 dialogue box.

### 6.5.5 "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.

### 6.5.6 "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 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-PLGW.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 (smarticons) for the menu items you can directly reach most of the program functions. The meaning of the smarticons 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.



**"Zoom out"**

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



**"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.



**"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.

### 6.5.7 "3D toolbar" menu item

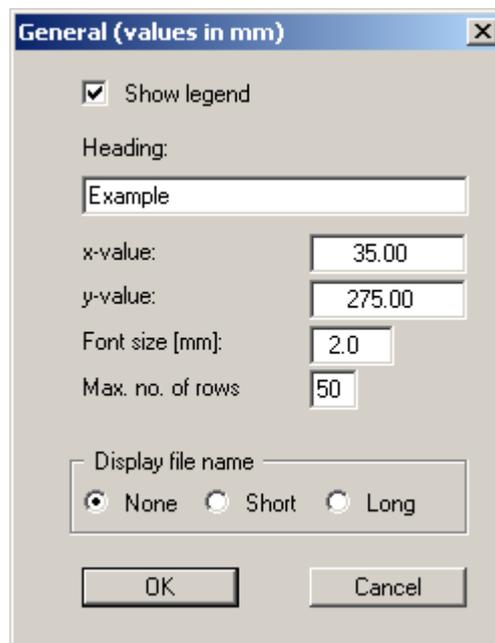
This menu item allows you to choose a pop-up window with tools for rotating and zooming three dimensional images.

The tools in this pop-up 3D window allow 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 (default: 45°)

can be adjusted as wished using the  tool.

### 6.5.8 "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.



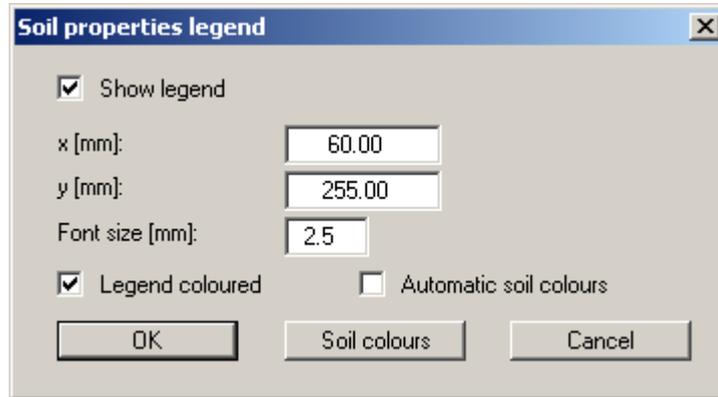
In addition to the heading, this legend contains information on the current graphics and preferences. For example, if you display the velocity field you will see information on the velocities scale.

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. 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 file. Any project identification entered (see Section 6.2.1) will also be shown in the general legend.

### 6.5.9 "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 not selected, the soil colours individually defined using the "**Soil colours**" button will be adopted.
- "**Soil colours**"  
You will see a dialogue 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 dialogue box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the "**Info**" button.

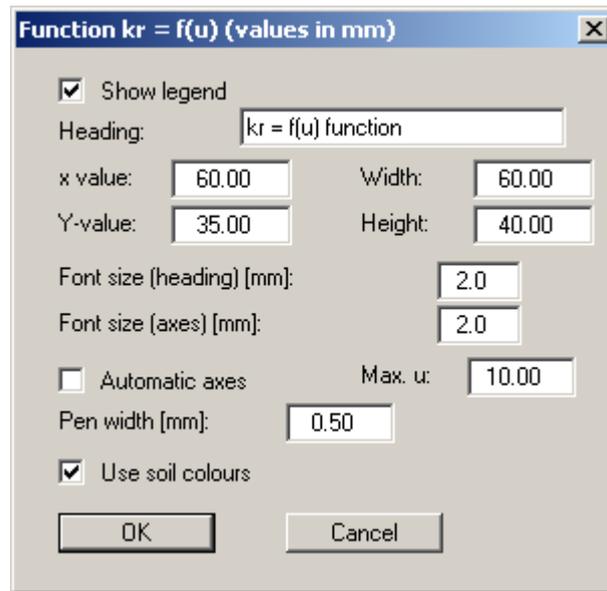
### 6.5.10 "Section legend" menu item

If you activate the "**Show legend**" check box in the dialogue 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.

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

The value  $k_r$  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 dialogue 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

### 6.5.12 "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 porewater pressure  $u$ . The legend shows the specific characteristic curves of the user-defined soils.

In the dialogue 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 Fehler! Verweisquelle konnte nicht gefunden werden.).

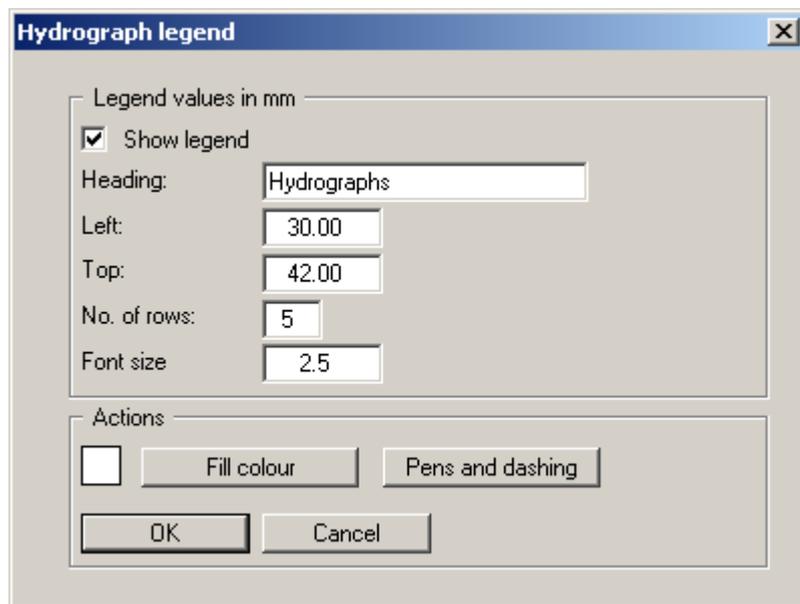
### 6.5.13 "k = f(t) function legend" menu item

If modelling was carried out in **GGU-TRANSIENT** using soil permeabilities varying as a function of time, the  $k = f(t)$  function is displayed as a graphic image when the "**Show legend**" check box is activated.

In the dialogue box to this menu item preferences for the visualisation can be edited (see menu item "**Graphics preferences/kr = f(u) function legend**", Section **Fehler! Verweisquelle konnte nicht gefunden werden.**).

### 6.5.14 "Hydrograph legend" menu item

If the "**Show legend**" check box is activated in the dialogue box, the lines used for the hydrographs are displayed in a legend, together with the corresponding node numbers.

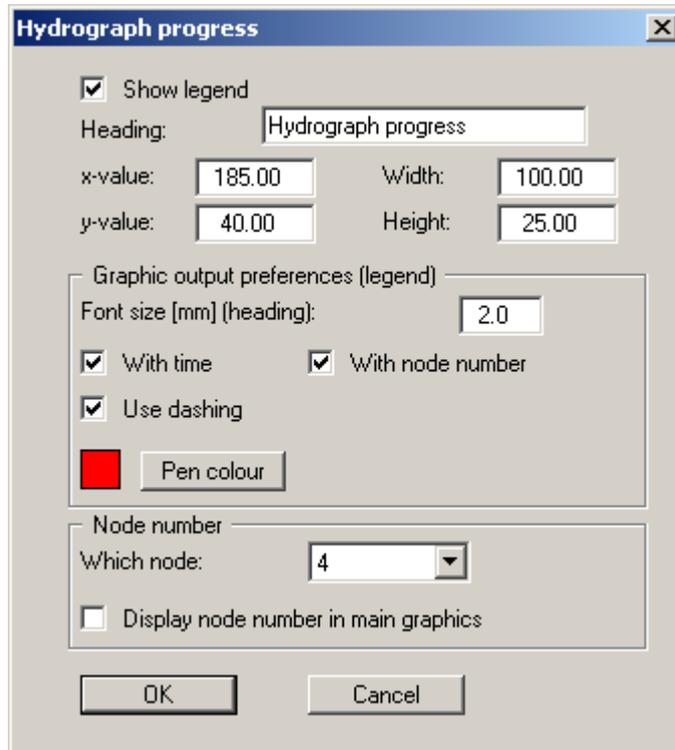


You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of rows**"; where necessary, several columns are used. The fastest way to modify the position of the legend is to press the [**F11**] function 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.

### 6.5.15 "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 "**Special**" menu).



Activate the "**Show legend**" check box and select the node from the lower group box of the dialogue 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.

### 6.5.16 "Move objects" menu item

Select this menu item in order to position legends 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.

### 6.5.17 "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-PLGW.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.

### 6.5.18 "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 re-freshed.

## 6.6 Page size + margins menu

---

### 6.6.1 "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 using "**Page size + margins/Manual resize (mouse)**" or "**Page size + margins/Manual resize (editor)**", you can quickly achieve a complete view using this menu item. This function can also be accessed using the **[F9]** function key.

### 6.6.2 "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.

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

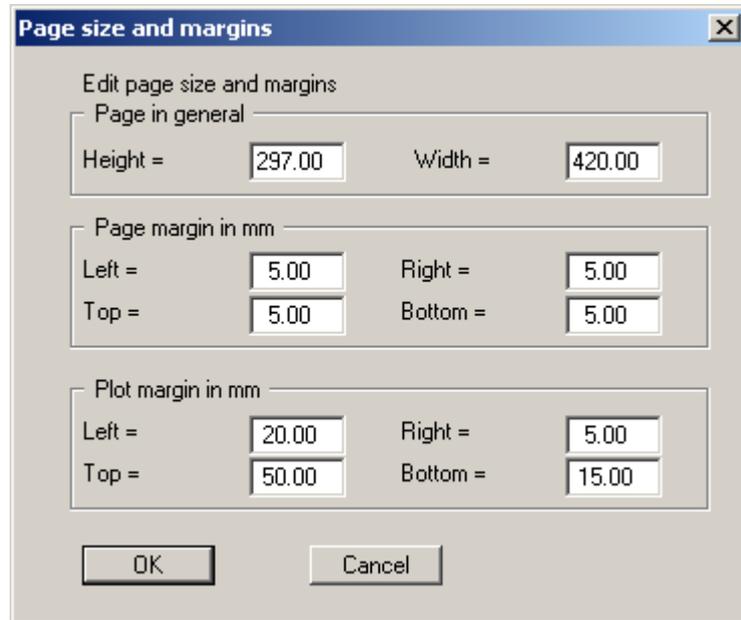
You can alter the image coordinates by direct numerical input in a dialogue box. This allows precise scale input. The coordinates refer to the *drawing area*. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 6.6.4).

The image coordinates entered here can be saved in a file with the extension "**.BXY**" and be re-loaded 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.

#### 6.6.4 "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 dialogue 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**" switch in the "**Page size + margins/Margins and borders**" menu item (see Section 6.6.6).
- "**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**" switch in the "**Page size + margins/Margins and borders**" menu item (see Section 6.6.6).
- 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.

#### 6.6.5 "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.

#### 6.6.6 "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**" switches.

## 6.7 Times menu

---

### 6.7.1 "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-TRANISENT**.

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

### 6.7.2 "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.

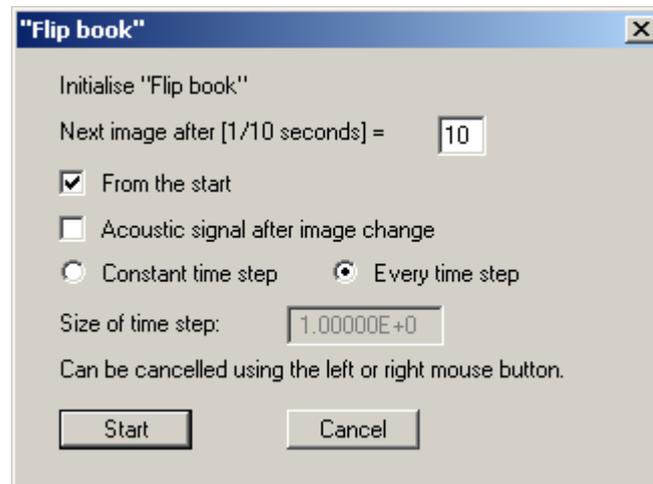
This function can also be called by pressing [F7] or by clicking the  icon in the menu toolbar.

### 6.7.3 "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.

#### 6.7.4 "Flip book" menu item

This is one of the program's most impressive types of evaluation. One of the menu items for contour line or velocities visualisation from the "Evaluation" menu must have been already started. Then go to this menu item to open the following dialogue box:



Here, you specify the speed of the animation, i.e. after how long a new image is to be displayed on the screen. Furthermore, you can activate an acoustic signal after each change in image. Finally, you specify the flow time after which the groundwater situation is to be displayed.

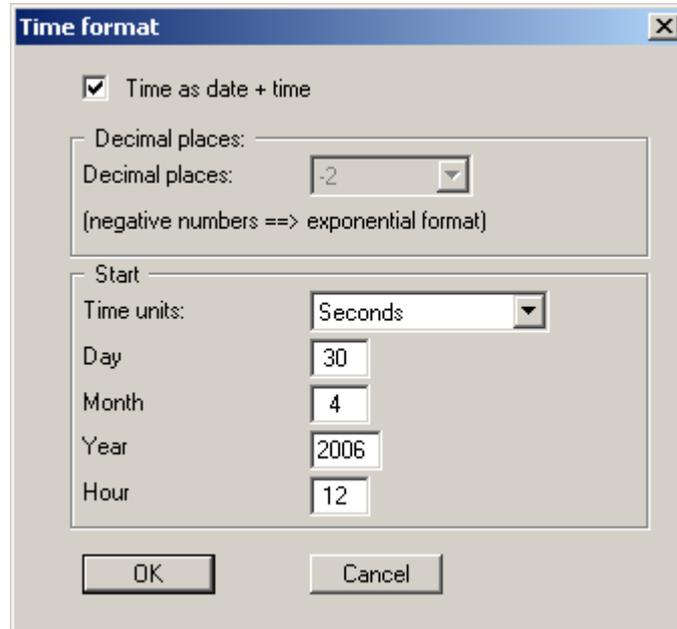
Using the "Start" command, you can begin the animation. If you would like to abort before the end time is reached, press any key.

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

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

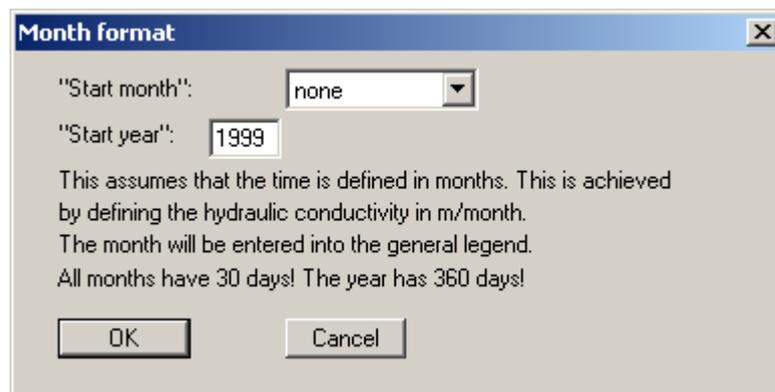
### 6.7.6 "Time format" menu item

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.



### 6.7.7 "Month format" menu item

For seasonally-dependent input values in particular, it makes sense to present the temporal course of events in a monthly format. Further explanations can be found in the corresponding dialogue box.



## 6.8 ? menu

---

### 6.8.1 "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-PLGW.

### 6.8.2 "Maxima" menu item

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

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

Using this menu item you can access the GGU Software website: [www.ggu-software.com](http://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>.

### 6.8.4 "GGU support" menu item

This menu item takes to the GGU-Software [Support area](#) at [www.ggu-software.com](http://www.ggu-software.com).

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

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

### 6.8.6 "Help" menu item

The GGU-PLGW online-help is opened using an installed browser (e.g. MS Internet Explorer). The help function can also be accessed using the [F1] function key.

### 6.8.7 "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 dialogues, menus)" und "Graphiktexte übersetzen (translate graphics)".

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

---

## 7 Tips

---

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

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.

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 online-help.
- [F2] refreshes the screen without altering the current magnification.
- [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 Index

---

### 5

50 day line ..... 22

### A

Angle of rotation, 3D graphics ..... 28  
ASCII file, coordinates FEM mesh..... 8  
ASCII file, hydrograph points ..... 24  
ASCII file, modelling results ..... 23

### B

Blanket layer discharge, as circle diagram..... 19

### C

Clipboard ..... 10  
Colour bar, define for contours ..... 14  
Colours + pens for graphical elements ..... 26  
Colours, define for soils ..... 29  
Colours, display in soil properties legend ..... 29  
Colours, switch on/off ..... 27  
Company letterhead, add via Mini-CAD ..... 27  
Confined GW areas, marking ..... 19  
Continuity condition ..... 18  
Contours potentials, 3D array graphics ..... 16  
Contours, dashing ..... 13  
Contours, groundwater thickness ..... 19  
Contours, groundwater-surface distance ..... 19  
Contours, layer boundaries ..... 20  
Contours, layer thickness ..... 20  
Coordinates, alter using an editor ..... 33  
Coordinates, alter using mouse ..... 33  
Coordinates, optimise ..... 33  
Coordinates, save/load ..... 33  
Copy/print area ..... 10, 27  
Course line method ..... 21  
Cutting borders, switch on/off ..... 34

### D

Differential record, generate ..... 8  
Discharge, display at FEM nodes ..... 18  
Discharges, for regions of the system ..... 18  
Drawing area ..... 33, 34  
DXF file, export ..... 9

### E

EMF format ..... 10  
Extraction well, consideration of ..... 22

### F

FEM mesh coordinates, export to ASCII file ..... 8  
FEM mesh, display outline ..... 12  
FEM mesh, maximum values ..... 38

FEM mesh, representation preferences ..... 12  
File name, display in legend ..... 28  
Flow lines, consider depressions ..... 22  
Flow lines, define/delete points ..... 21  
Flow lines, display ..... 21  
Flow lines, with distance increment ..... 22  
Flow lines, with time increment ..... 22  
Flow variables, display as hydrographs ..... 23  
Flow variables, display in section ..... 25  
Font size, drawing elements ..... 34  
Font size, general legend ..... 28  
Font size,  $kr = f(u)$  legend ..... 30  
Font size, soil properties legend ..... 29  
Function  $k = f(t)$ , display legend ..... 31  
Function keys ..... 40  
Function  $kr = f(u)$ , display legend ..... 30  
Function  $nw = f(u)$ , display legend ..... 30

### G

General page informations, add via  
    Mini-CAD ..... 27  
GGUCAD file, export ..... 9  
GGUMiniCAD file, export ..... 10  
GGU-STABILITY ..... 8  
Gradients, display ..... 17  
Graphics, add via Mini-CAD ..... 27  
Groundwater thickness, as contours ..... 19  
Groundwater-filled volume ..... 19  
Groundwater-surface distance, as contours ..... 19

### H

Header CAD, use ..... 27  
Hydrographs, define/delete nodes ..... 23  
Hydrographs, display ..... 24  
Hydrographs, save/load nodes ..... 24  
Hydrographs, show nodes ..... 24

### I

Installation ..... 5

### K

Knowledge Base ..... 38

### L

Language preferences ..... 5, 38  
Layer boundaries, as contours ..... 20  
Layer thickness, as contours ..... 20  
Layout, output sheet ..... 34  
Legend, hydrograph progress ..... 32  
Legend, hydrographs ..... 31  
Legend, section course ..... 29  
Legend, soil properties ..... 29  
Legend, system properties ..... 28  
Legends, move with mouse ..... 32

Licence protection ..... 5

## M

Metafile, export ..... 10  
Mini-CAD file, export ..... 10  
Mini-CAD, use ..... 27  
Modelling results, display for individual  
    FEM nodes ..... 23  
Modelling results, export to ASCII file ..... 23  
Mouse wheel functions ..... 39

## P

Page format, define ..... 34  
Page margins, define ..... 34  
Page margins, switch on/off ..... 34  
Particle-tracking method ..... 21  
Pen preferences ..... 26  
Permeability, define as function of time ..... 31  
Phreatic lines, display ..... 25  
Plot margins, define ..... 34  
Porewater pressure mesh, for slope stability  
    program ..... 8  
Potential contours, 3D graphics ..... 14  
Potential contours, colour visualisation ..... 13  
Potential contours, normal visualisation ..... 12  
Print, graphics ..... 9  
Print, section ..... 10  
Print, several files ..... 11  
Printer preferences ..... 9  
Program, improvements ..... 38  
Program, information ..... 38  
Program, preferences ..... 32  
Project data, add via Mini-CAD ..... 27  
Project identification, display ..... 28  
Project identification, enter ..... 12

## R

Record description, display ..... 28

Record description, enter ..... 12

## S

Scale, define ..... 33  
Scroll the screen ..... 39  
Section course, define ..... 25  
Section course, display ..... 25  
Section course, save/load ..... 25  
Seepage velocity, display ..... 17  
Smarticons, for menu items ..... 27  
Smarticons, of 3D graphics ..... 28  
Smoothing out procedures, contours ..... 13  
Soils, define colours ..... 29  
Status bar main program, activate ..... 27  
System, information ..... 38

## T

Time format, define ..... 37  
Time format, define seasonally-dependent ..... 37  
Time step, select for representation ..... 35  
Toolbar, for menu items ..... 27  
Toolbar, of 3D graphics ..... 28  
Translation ..... 38  
True-type font ..... 26

## V

Velocity field, display ..... 17

## W

What you see is what you get ..... 26

## Z

Zoom factor, define for full-screen display ..... 26  
Zoom function ..... 26, 39