Modelling of contamination propagation in groundwater systems using the finite element method
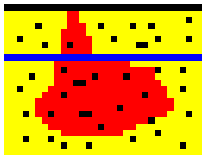
GGU-CONTAM-FE

VERSION 5

Contents:

1 Preface

The **GGU-CONTAM-FE** program allows modelling of contaminant propagation in horizontal-plane, vertical-plane and axis-symmetrical groundwater systems using the finite-element method. A record containing the basic steady-state data must be previously generated using the **GGU-SS-FLOW2D** program.

The **GGU-PLGW** program is available for evaluation and visualisation of modelling results. It enables impressive, time-dependent animation or presentation of concentrations along sections through the system.

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

It is not the task of this manual to give an introduction to contaminant propagation analysis using finite-element methods. For details, please refer to the literature.

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 finite-element programs). 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 forming the basis for contaminant transport modelling are records from steady-state modelling using **GGU-SS-FLOW2D**.

- Triangular elements are used.

- Concentrations are computed at every node. Concentration distribution is linear within the triangular elements.

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-CONTAM-FE** 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-CONTAM-FE** 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-CONTAM-FE 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

The flow field is required in order to model contaminant propagation (velocities with magnitude and direction). **GGU-CONTAM-FE** acquires this initial condition from a record (e.g. "**plgw.da1**") generated during a previous modelling using the **GGU-SS FLOW2D** application. It is therefore necessary to load this file first after opening the program. The flow field contained in this file remains constant for subsequent calculations.

Besides the flow field, the **GGU-CONTAM-FE** program requires information on the initial contaminant distribution conditions at time t = 0. These so-called initial concentrations can be arbitrarily defined. If you do not define any values, the program will assume an initial concentration of '**0**' at all nodes.

Apart from the flow field, the "**plgw.da1**" example file also includes the effective pore space neff, upon which steady-state modelling is based. This *steady-state* soil property (n_{eff}) cannot be altered from within the **GGU-CONTAM-FE** program. If amendment of this value were allowed, the flow field would no longer correlate to the computed potentials and contaminant propagation modelling would produce nonsensical results. However, additional soil properties, which are required for contaminant propagation, must be provided.

Further to these additional soil properties, time-dependent boundary conditions can also be defined with **GGU-CONTAM-FE**. You must define boundary values (concentration or concentration source) 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 (for contaminant propagation) and polygon courses, as well as their allocations to system nodes, can be saved in a separate file (known as boundary data), in order to make them available again for later modelling.

Generally, when modelling contaminant propagation, a flood of result values is generated. Modelling results are saved by the **GGU-CONTAM-FE** program in a file ("***_kon.plw**"). Contaminant propagation data cannot be evaluated using the **GGU-CONTAM-FE** program. The **GGU-PLGW** program is available for evaluation purposes and is described in a separate user-manual.

The **GGU-CONTAM-FE** 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, if only to save renewed input in the case of a power cut.

5   Starting the program

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

- File

- ?

After clicking on the "**File**" menu, a system processed with the **GGU-SS-FLOW2D** program can be loaded by means of the "**Load steady-state 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-CONTAM-FE**. 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 seven menus:

- File

- Editor

- Boundary values

- 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-CONTAM-FE** screen is not refreshed after every alteration.

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

6   Short description

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

- In order to be able to use the system entered in your **GGU-SS-FLOW2D** groundwater flow program for contaminant propagation modelling using the **GGU-CONTAM-FE** 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-CONTAM-FE** program and using the menu item "**File/Load steady-state GW data**", select the desired record of a flow system.

- Using the "**Editor**" menu you can define your "**Soil properties**". A description of the individual constants can be found in the "**Info**" function field.

- Using the "**Editor/Boundary condition polygons**" menu item, you specify the temporal changes in concentration as polygon courses. This is initially independent of location. By clicking on the menu item you will see a dialogue box within which, by clicking the "**New**" field and then "**Edit polygon course**", you can specify the number of polygon points, i.e. the number of temporal changes in your concentrations.

- The initial system concentration is specified via the menu item "**Boundary values/Individual initial concentrations**". It is possible to set the concentration individually for specific nodes, or for several nodes in section.

- Contaminant propagation modelling is started by selecting "**Modelling/Start**". Specify the desired end time at which the calculations are to cease. Subsequently, checks are carried out by the program (Peclet and Courant criteria). The Courant criterion determines the maximum permissible time step for your system. Before actual analysis begins, you will see this value in a dialogue box. You can edit this value. The Peclet criterion principally concerns the element size; if this criterion is not satisfied, you will generally have no other choice but to analyse with a finer mesh (**GGU-SS-FLOW2D**).

- 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-CONTAM-FE** ("***_kon.plw**") into **GGU-PLGW**.

7 Contaminant propagation theory

7.1 Overview

The primary aim of contaminant propagation modelling is the prognosis of future conditions on the basis of given initial data and, on occasion, pre-specified counter-measures.

Using a numerical analysis approach, even complicated systems can be investigated and assessed. Precise modelling of the natural conditions is difficult, due to the multitude of initial parameters required and the inhomogeneity of aquifers. Before any contaminant analysis, you should therefore ensure that your initial data justify this.

7.2 Mechanisms relevant to contaminant propagation

Five important mechanisms describe contaminant propagation in groundwater and must be entered as initial data for propagation simulation:

- **Convection**
  is the migration of contaminants in the direction, and with the apparent flow velocity, of groundwater flow. This propagation mechanism is normally dominant in a flowing system, but is superimposed upon and influenced by others. Mathematically, this mechanism enters the differential equation in the shape of the apparent flow velocities u_x and u_y.

- **Dispersion**
  includes the local deviation from the mean flow velocity. One differentiates between dispersion caused by the grain skeleton, which describes, amongst other factors, deviations in the flow course around the grain skeleton, and macro-dispersion, which is the dominant mechanism in wide-area contaminant propagation, e.g. in sand lenses and layering. This mechanism is described by the mathematical values of longitudinal (in flow direction) dispersivity α_L (m) and transverse (perpendicular to flow direction) dispersivity α_T (m).

- **Diffusion**
  This type of propagation is determined by the concentration gradient. It is normally only effective in systems with extremely low velocity. In water at 10 °C, the diffusion coefficient Dm is 10^{-9} m/s².

- **Adsorption**
  expresses the inhibition of contaminant migration due to adsorption onto the grain skeleton and is described by the sorption constant sorp (-). The sorption constant is the product of grain density and the adsorption coefficient. Contaminant propagation is retarded by adsorption.

- **Decay**
  As a result of physical, chemical and biological reworking, contaminants are eliminated from the groundwater. This process is described by the decay constant λ (l/s).

A mass balance equation and a space- and time-dependent, analysable differential equation results from these values:

Mass balance:

Increase in dissolved contaminant mass =

> net input by convection
> + net input by diffusion and dispersion
> + input from contaminant sources
> - extraction from wells
> - loss through decay reactions
> - adsorption onto the grain matrix

7.3   Contaminant propagation modelling with FEM models

7.3.1   The Peclet criterion

As you know, FEM allows almost unlimited subdivision of the system into triangular elements. Concentrations are assumed to be linearly distributed across each element. Where the contaminant front is discrete, such an approach is no longer capable of precisely modelling contaminant distribution. The contaminant fronts then become *smeared*. This effect is known as numerical dispersion. It occurs predominantly when the proportion of convection is large compared to other mechanisms (dispersion and diffusion). This can be avoided by selecting a denser triangular mesh.

The "**Peclet criterion**" serves to check whether a system displays a tendency towards numerical dispersion and thus to compute the necessary mesh density. This criterion is checked by the program after modelling has begun. If it is exceeded, mesh refinement will be required. You will then need to reprocess the system using **GGU-SS-FLOW2D**. Under certain conditions, however, a very dense mesh is required. If necessary, you can obtain a solution using the RANDOM-WALK method, as this procedure does not recognise the numerical dispersion effect (**GGU-CONTAM-RW** program).

7.3.2   The Courant criterion

The precision of all numerical procedures is influenced by the time step Dt. Precise results can be achieved with small time steps, but analysis time increases correspondingly. The optimum time step size can be derived from the "**Courant criterion**".

After starting modelling, the program examines the Courant criterion and suggests the determined time step.

8   Description of menu items

8.1   File menu

8.1.1   "Load steady-state GW data" menu item

Import the data from a steady-sate groundwater model previously analysed using the **GGU-SS-FLOW2D** program using this menu item to form the basis for your contaminant transport modelling. 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.

8.1.2   "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.

8.1.3   "Load boundary values" menu item

Using this menu item you can reload a 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 contaminant transport modelling. The file contains the soil properties and the polygon courses, as well as the allocations of the polygon courses to the system nodes.

8.1.4   "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 with the suffix "**.kon_gw**"contains the soil properties and the polygon courses, as well as the allocations of the polygon courses to the system nodes.

8.1.5   "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.

8.1.6 "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 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.
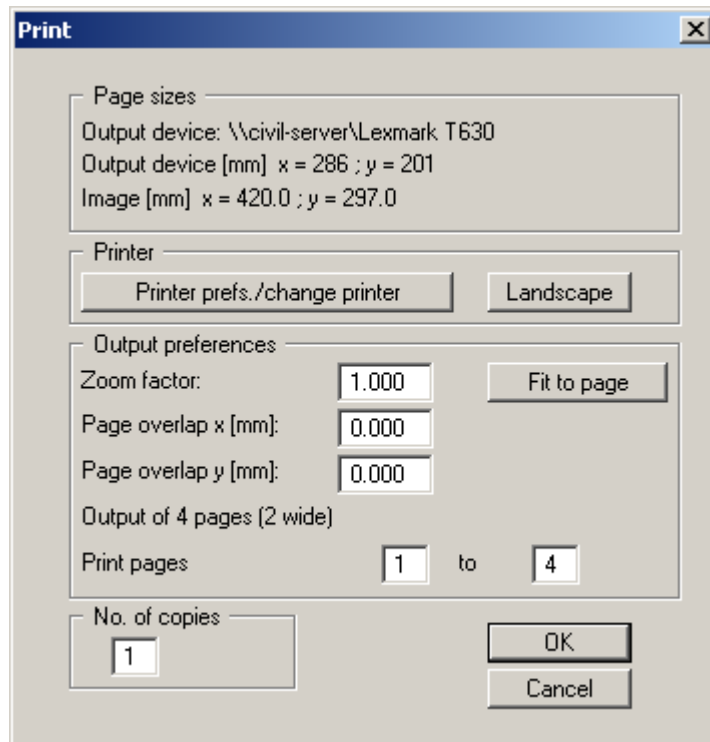
In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate 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 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.

8.1.7   "Exit" menu item

After a confirmation prompt, you can quit the program.

8.1.8   "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.

8.2.1   "Soil properties" menu item

After going to this menu item a system-specific dialogue box opens allowing input of the soil properties. The *steady-state* property n_{eff} imported from the **GGU-SS-FLOW2D** program can no longer be amended here

In the example above, three soils have been defined for the system (groundwater modelling with **GGU-SS-FLOW2D**). Enter the soil parameters relevant to contaminant transport in the input boxes:

- **"Info"**
  This menu item provides some information on the meaning of the properties and their units. Take care to always use the same dimensions, as these are not pre-specified by the program. The result will be given in terms of the dimensions with which you entered your data.

- **"alph_l"**
  Enter the value of the longitudinal dispersivity alph_l. This is a function of grain size and varies between a few centimetres (laboratory tests in sand) and over 100 metres (jointed rocks).

- **"alph_t"**
  The transversal dispersivity alph_t is generally a magnitude smaller than the longitudinal. In field tests, ratios around 0.01 were determined. Values of up to 0.3 are given in the literature.

- **"Dm"**
  Enter the value of the diffusion coefficient Dm. This value only influences contaminant distribution at very low flow velocities. For water at 10°C, this value is 10^{-9} m/s².

- **"Sorp"**
  Enter the value of the sorption constant sorp. The sorption constant is dimensionless and is the product of grain density and the adsorption coefficient.

- **"lambda"**
  Enter the value of the decay constant lambda.

8.2.2 "Boundary condition polygons" menu item

The temporally variable system boundary conditions for the system are defined using this menu item, or previously defined polygon courses may be edited. You enter the concentrations for the specified time intervals. Input is initially location-independent. After clicking the numbered button of an existing polygon course or the "**New**" button, a dialogue box opens providing the following options:

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

- "**Edit polygon course**"
  The following dialogue box appears for example:

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. **GGU-CONTAM-FE** works with true dimensions. Times must therefore be given in the time dimension used for permeability. For example, if permeability was entered in m/s, times must be entered in seconds. The polygon value must be entered in mass/time (e.g. mg/s) for concentration sources, or mass/length³ for concentrations. The length units must also correspond to the permeability length units.

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

8.3.1   "Individual initial concentrations" menu item

This menu item at first generates a dialogue box in which you specify the number of decimal points for concentrations, and the font size.

If you confirm with "**OK**", the FEM mesh will be displayed on the screen. Click on the appropriate point with the left mouse button. The current system coordinates of the mouse pointer are shown in the status bar at the bottom of the screen. Enter the desired initial concentration in the following dialogue box.
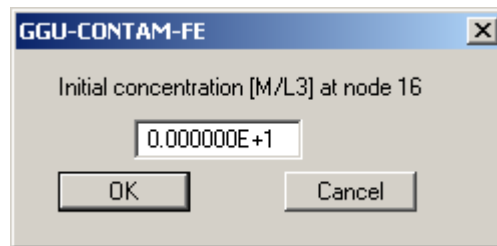
The length dimension must also coincide with the permeability length dimension. If you click on a node with the right mouse button, any current initial concentration will be deleted and the node set to '**0**'.

8.3.2   "(Initial concentrations) In section" menu item

Click on four points in a counter-clockwise direction to define the desired section. You will then be asked for the initial concentration you would like to apply to all nodes in the section. Alternatively, you have the possibility of deleting all initial concentrations, i.e. setting them all to '**0**'.

8.3.3   "(Initial concentrations) From file" menu item

You can load concentrations from a previous analysis as initial concentrations. The first time step after the given time will be read.

8.3.4 "Individual boundary conditions" menu item

After asking for the font size, this menu item allows allocation of a boundary condition polygon to a specific node.

Once you have clicked on a node with the left mouse button, the following dialogue box appears:
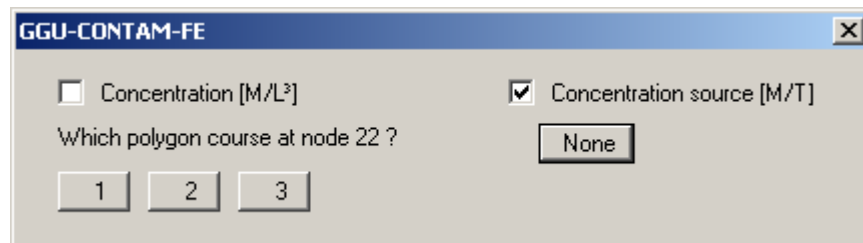
In the example above you can select from three pre-defined boundary condition polygons. Further, you can specify whether the boundary condition is to be a concentration source or a concentration.

If you click on a node with the right mouse button, any current polygon allocation will be deleted.

8.3.5 "(Boundary conditions) In section" menu item

By selecting this menu item you can assign a boundary condition polygon to a system section. By means of an info dialogue box, you can define a section with the left mouse button, which you can then assign a previously specified polygon course after being asked if you would like to set or delete a condition.

8.4.1 "Display mesh" menu item

With this menu item you can choose to display the groundwater flow model without any labelling.

8.4.2 "Start" menu item

After clicking on this menu item or, alternatively, by pressing [**F5**], several dialogue boxes appear with information and queries about the Peclet and Courant criteria (*cf*. Section 7), which you may accept or edit. The following dialogue box then appears:
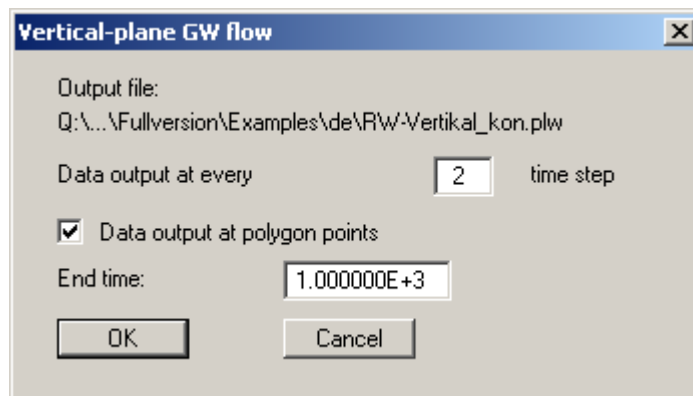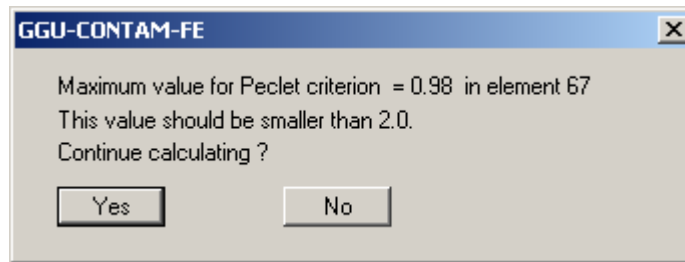

The example shown is for a vertical-plane system. You will see the name of the output file (here "**RW-Vertikal_kon.plw**"). When calculations are complete, this file contains all result data, which can then be evaluated with **GGU-PLGW**.

A flood of results is usually provided for contaminant propagation analyses, with correspondingly large result files. Often, a small time step must be selected in order to achieve sufficiently precise results. You can therefore specify the times at which data output should follow.

Independently of the selected time step, the program automatically proceeds to all times associated with boundary polygons. If you activate the "**Data output at polygon points**" switch, the results for these times will be issued additionally, independent of the previous entry. It is conceivable, for example, to enter a very large value for "**Data output every nth time step**" and to temporally subdivide a boundary condition polygon such that output only occurs at the times specified by you.

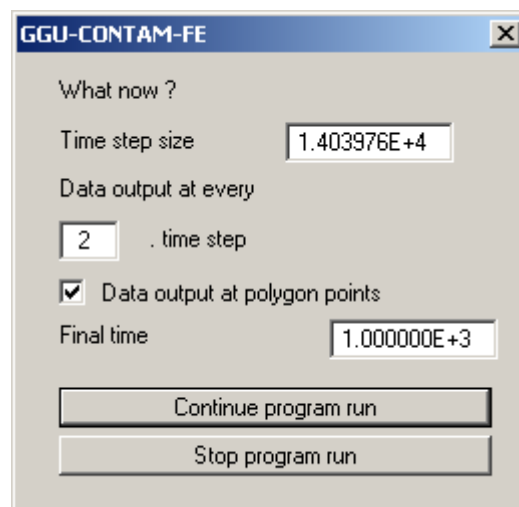Finally, you enter the end time at which analysis is to cease.

After selecting the "**OK**" button, the program carries out checks. First, the Peclet criterion is checked.

GGU-CONTAM-FE

Maximum value for Peclet criterion = 0.98 in element 67
This value should be smaller than 2.0.
Continue calculating ?

[Yes]   [No]

The Courant criterion is then checked and the optimum time step shown.

GGU-CONTAM-FE

Maximum time step for Courant criterion = 1.40397E+4
With what time step do you want to calculate?

1.403976E+4
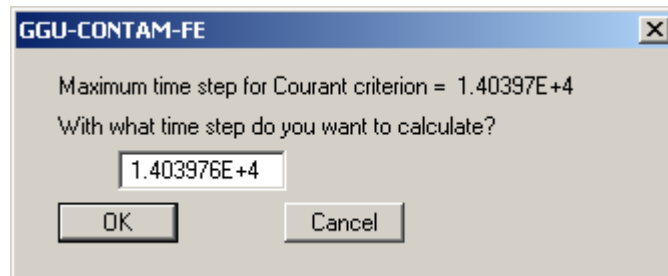
[OK]   [Cancel]

After selecting the "**OK**" button the calculation proper will begin. A dialogue box with a "**Cancel**" button will inform you of analysis progress. If you select this button, analysis will not cease immediately, but instead you can edit specific program data and then carry on calculating.

GGU-CONTAM-FE

What now ?

Time step size   1.403976E+4

Data output at every

2   . time step

☑ Data output at polygon points

Final time   1.000000E+3

[Continue program run]
[Stop program run]

A record with the extension "***_kon.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.

8.5.1 "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.
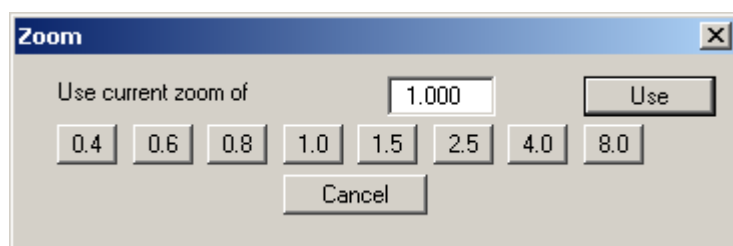
A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "**Use**" to exit the box the current factor is accepted. By clicking on the "**0.4**", "**0.6**", etc. buttons, the selected factor is used directly and the dialogue box closed.

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.

8.5.2 "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.

8.5.3 "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can edit the pen settings for the initial concentration and boundary conditions. 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.

8.5.4 "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.

8.5.5 "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.

8.6 Page size + margins menu

8.6.1 "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.

8.6.2 "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.

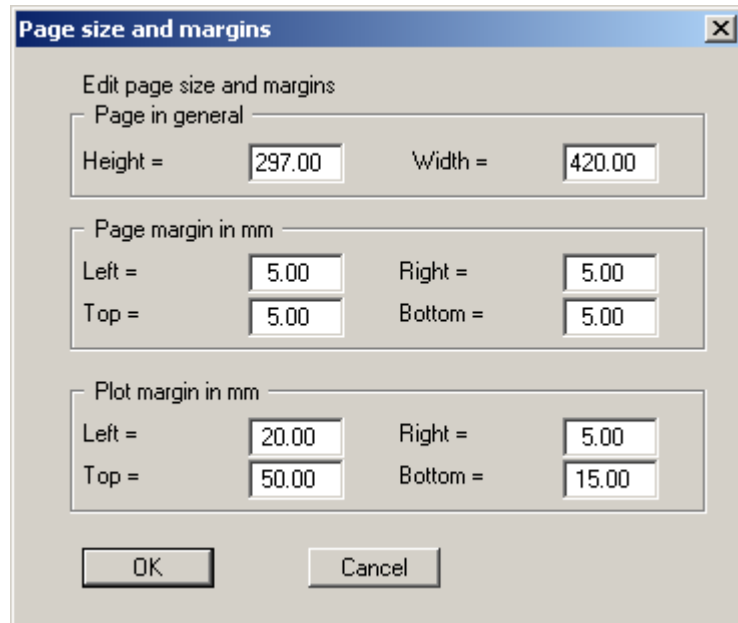### 8.6.3 "Manual resize (editor)" menu item

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 8.6.5).

8.6.4 "Font size selection" menu item

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

8.6.5 "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 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.

- "**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.

8.7.1   "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-CONTAM-FE**.

8.7.2   "Maxima" menu item

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

8.7.3   "Help" menu item

The **GGU-CONTAM-FE** 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.

8.7.4   "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.

8.7.5   "GGU support" menu item

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

8.7.6   "What's new?" menu item

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

8.7.7   "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 switches "**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.

9 Tips

Slow screen output generated using the contour line menu items can be cancelled by clicking the right mouse button.

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 object 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.
- [**F5**] opens the menu item "**Modelling/Start**".
- [**F9**] activates the menu item "**Page size** + **margins/Auto-resize**".

10 Index