triwaco

groundwater modelling software

4 Setting up a groundwater model

ROYAL HASKONING

Chapter 4: Setting up a groundwater model

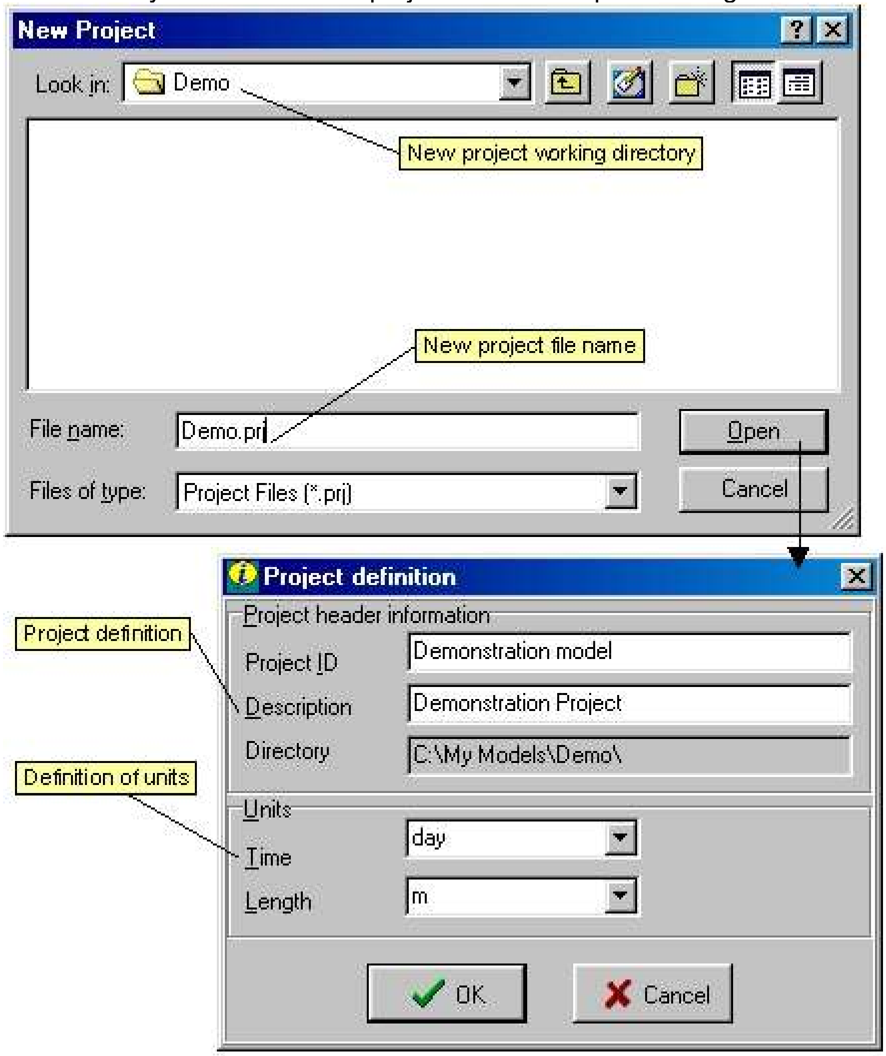
4.1 Introduction..4-3
4.2 Creating a Project..4-3
4.3 Creating a Grid data set..4-6
4.3.1 Introduction..4-6
4.3.2 Opening a Grid data set.......................................4-6
4.3.3 Defining Grid parameters......................................4-7
4.3.4 Generating the Grid...4-10
4.3.5 Viewing the Grid..4-10
4.3.6 Input data description..4-11
4.3.7 Output data description.......................................4-15
4.3.8 Alternative grid generators...................................4-16
4.4 Creating an Initial data set.....................................4-17
4.4.1 Introduction..4-17
4.4.2 Opening an Initial data set...................................4-17
4.4.3 Defining model properties.....................................4-17
4.4.4 Defining model parameters (general)...........................4-20
4.4.5 Definition of boundary conditions.............................4-22
4.4.6 Definition of river (line-element) parameters.................4-22
4.4.7 Definition of source parameters...............................4-23
4.4.8 Definition of hydrogeological parameters......................4-24
4.4.9 Definition of anisotropy......................................4-24
4.4.10 Definition of expressions.....................................4-25

4.1 Introduction

A model in **Triwaco** is always set up as a project. The first step of setting up a model is by defining the conceptual model. The conceptual model is created in the Initial Set, containing all parameter maps, maps that are independent of the grid. These parameter maps can be edited spatially using DigEdit, or directly be imported from a GIS. Next step is creating the appropriate grid. **Triwaco** allows the user to calculate groundwater flow with a Finite Element Grid (Flairs) or a Finite Difference Grid (Modflow). The allocation of parameter values to the grid and the simulation of groundwater flow is explained in the next chapter 5.

4.2 Creating a Project

Issuing the command 'New', or pressing the  -icon-button on the menu bar, will cause the program to ask the user to define the directory and name of the project file and to open a dialog box.

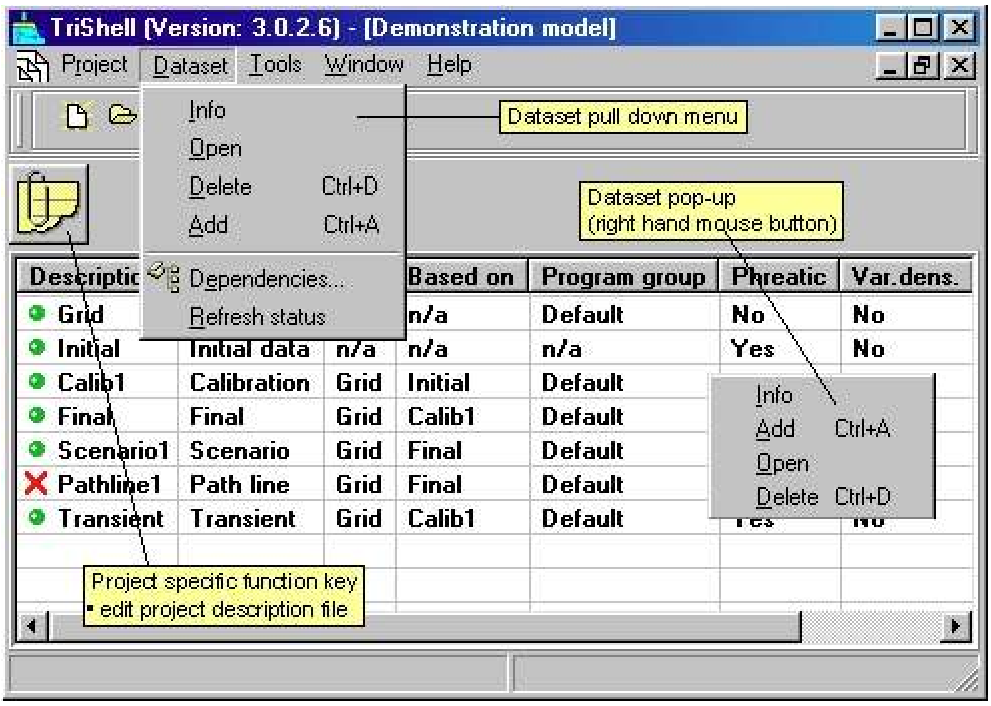


The user has to provide the appropriate information, which consists of a Project ID, a Description and the project's working directory. Moreover, the user may specify a different set of units (by default **Triwaco** uses the time units **day** and the length unit **meter**). The definition of all parameters has to be in correspondence with these units.

Pressing the OK-button the program will create a project file and open the project window. If the user selects an existing project file he will be prompted whether or not to overwrite this file. An (empty) project window will be opened and in the menu bar the following two menu options are added: 'data set' and 'Window', each having their own pull down menu.

The project window contains an additional, project specific, function key.

 Start the default text-editing tool and opens a project description file (project-name.dsc).

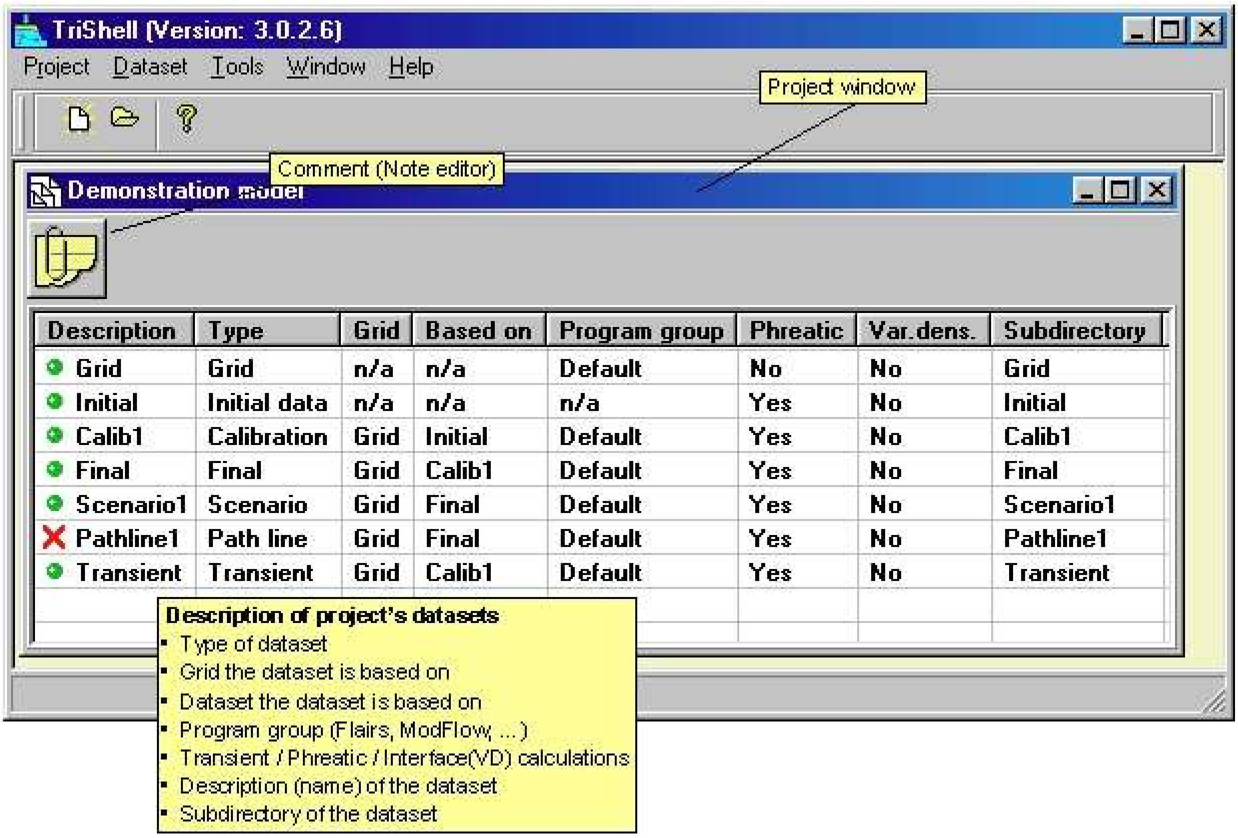


The 'data set' pull down menu allows the user to add, open and delete data sets to and from the project window. These functions, which appear in the upper part of the pull down menu, can also be accessed by the data set pop-up menu that is activated by the right-hand mouse button.

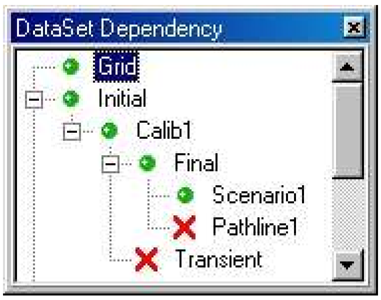
Data set pull down menu		**Description**
Info		Display information on the opened data set
Open		Open an existing data set
Delete	Ctrl D	Delete the selected data set
Add	Ctrl A	Add a new data set to the project
Dependencies…		Display data set dependency window
Refresh status		Check and update status indicator

Every data set will be added to the project window of the current modelling project. The project window displays the following information.

Type	**Description of type of data set.**
Grid	Finite Element or Finite Difference grid to be used for calculations
Based on	Data set with (hydrogeological) parameters to which this set refers
Transient	Indicates whether or not transient calculations are carried out
Phreatic	Indicates whether or not the uppermost aquifer is phreatic
Var. dens.	Indicates whether or not the variable density module is used
Description	Descriptive commentary of the data set and its use
Subdirectory	Name of the subdirectory for this data set



The functions in the lower part of the data set pull down menu allow the user to check the dependencies between the various data sets of one project and to refresh the status indicators. Selecting 'Dependencies…' from the pull down menu displays the data set Dependency window.



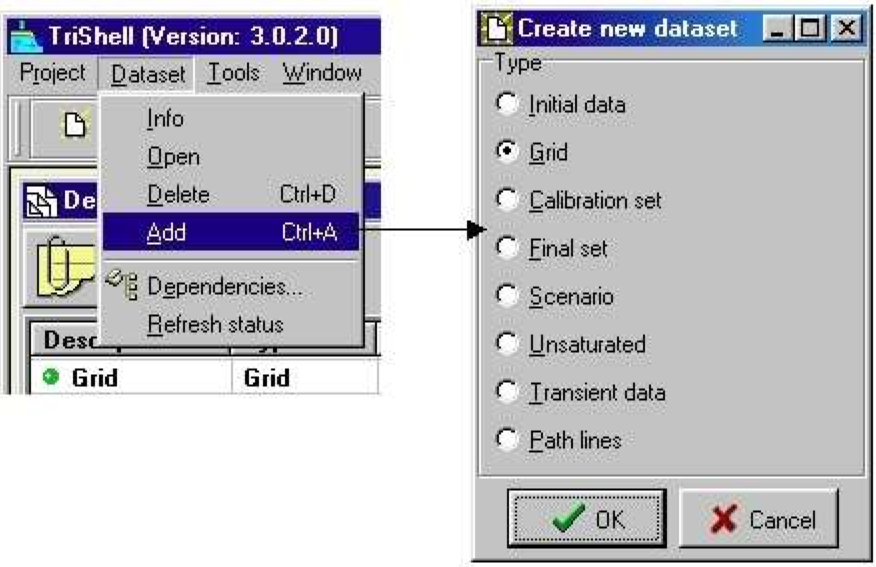
4.3 Creating a Grid data set

4.3.1 Introduction

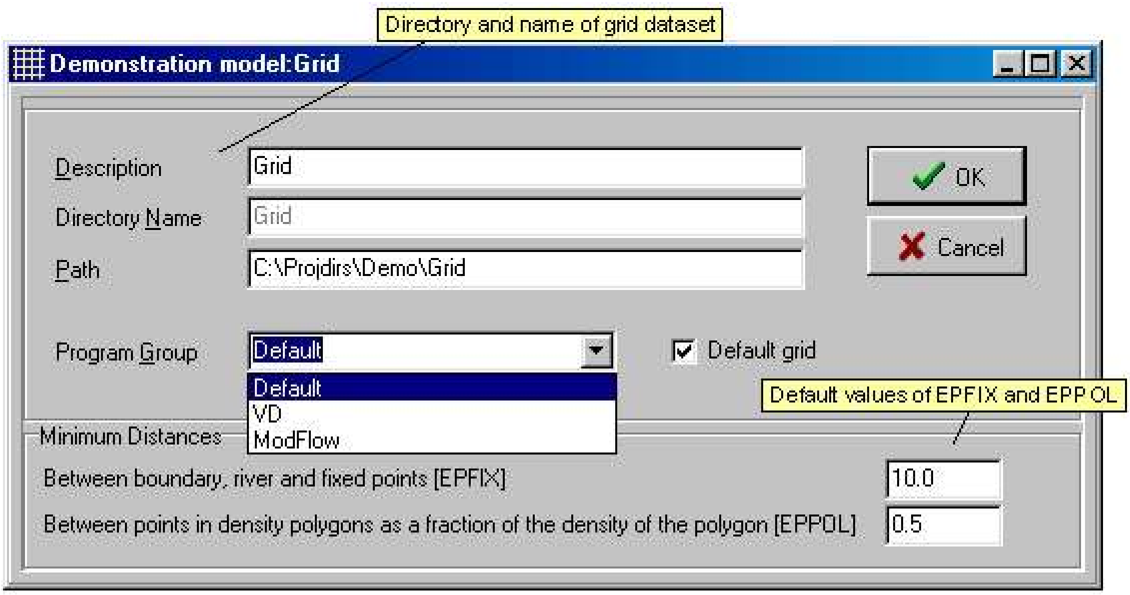
Triwaco can handle two types of grids, Finite Element Grids and Finite Difference Grids. For the Finite Element Grid **Triwaco** uses **Triwaco-Flairs** for calulating groundwater flow and the grid generator program **Tesnet.** For Finite Difference Grid **Triwaco** uses MODFLOW-96 of the USGS and the grid generator program **Monet**. Once the program group (and hence the grid and simulation program) is defined in the 'Grid definition'- window, the **TriShell** processes the initial data, keeps track of changes in data(sets), runs the corresponding separate modules and carries out different simulation runs.

4.3.2 Opening a Grid data set

To create a grid the users opens the grid data set selecting 'data set' 'Add' from the pull down menu and selecting 'Grid' from the 'create new data set' dialog window.



The program now displays the Grid data set info window and the user supplies the data set name and directory (if different from the name) and may change the default values for EPFIX and EPPOL. Marking the section 'Default Grid' with a ☑ the data set's grid will be used whenever the graphical presentation tool Triplot is started selecting the function key in the project window's title bar.



Choosing the 'Program group' allows the user to calculate with the Finite Element Grid (Flairs) by selecting 'default' or calculate with the Finite Difference Grid (Modflow) by selecting 'ModFlow'. When a model is created using the Variable Density option choose 'Variable Density' (not available in the Standard Package, see also Chapter 14). The restrictions on using the Finite Difference grid for ModFlow are described in chapter 5.

The parameters EPFIX and EPPOL define the minimum distances between nodal points of the Finite Element Grid to be maintained during generation of the grid (valid only for Finite Elements):

- **EPFIX** : Minimal distance between 'Fixed points', e.g. points defined as vertices of the boundary and the rivers or as sources.
- **EPPOL** : Minimal distance of points within a density polygon, expressed as fraction of the nominal distance defined for the polygon.

After definition of the grid properties the grid data set is added to the project window. Opening the grid data set displays the grid data set window, containing the grid parameters: BND, POL, RIV and SRC. In the data set window's title bar the description file function key appears which allows the user to add comments in a text file (model.dsc). EPFIX may be defined by polygons to vary EPFIX. EPFIX should be defined by the parameter name EPFIX.

4.3.3 Defining Grid parameters

The input data for generation of the grid consist of the following items, which are defined by the parameters from the grid data set:

- The model area, defined by the grid ***boundary***. The boundary of the model will be defined by the corner points of a polygon (the parameter **BND** of the ***grid data set***). The number of nodes to be generated on each boundary segment can either be specified by the user (by ***editing*** the ***input file***) or be generated by the program using the node distances specified in the ***density polygon*** map.

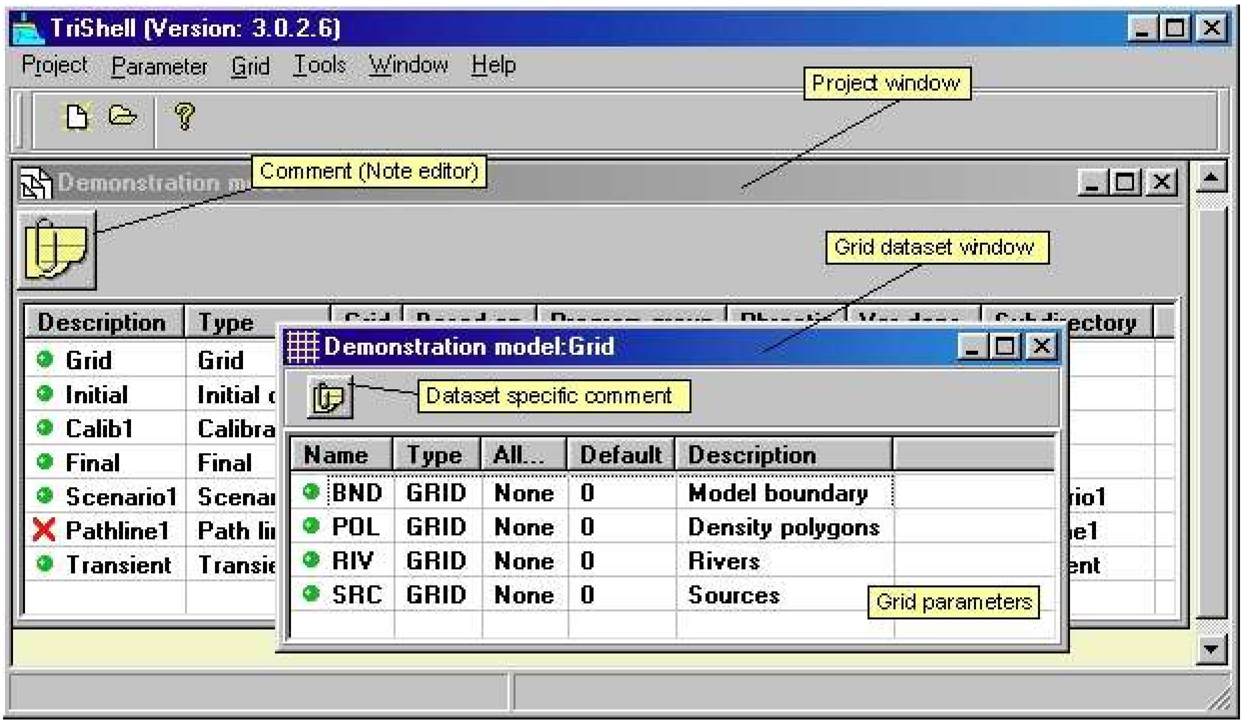
- The ***rivers (line elements)***, strings of line segments between river points. The input for a river contains the river points (the parameter **RIV** of the ***grid data set***). The river segments between the points are straight. The number of nodes that will be placed on a segment can be specified by the user (by ***editing*** the ***input file***) or calculated by the program using the node distances specified in the ***density polygon*** map. The numbering of the rivers (defined in the corresponding map file) is not necessarily sequential. The only demand, regarding the numbering of the rivers, is that each river has a **unique ID,** defined in the par file. Rivers may be defined by a single line or by a number of parallel lines. In the latter case some additional ***editing*** of the ***input file*** is required. Other line-shaped elements (a mountainside or fracture zone) may be incorporated in the grid by defining a river with an ID equal to 0.

- The ***source*** points (fixed-point), which can be surrounded by support circles defining small elements. The location of each source is specified in the input (the parameter **SRC** of the ***grid data set***). The source nodes will be marked in the generated grid, but do not necessarily have to act as a source or sink, they can also be put at the location of monitoring wells, to get more accurate results than by interpolation between the three nodes of an element. The user can specify extra nodes to be created around the sources (Only for Finite Element grid, NOT for Finite Difference grid). The extra nodes will be located at concentric circles around the fixed point. The radii of the circles and the number of nodes on each circle are specified by the user (Select 'Grid' 'Define support circles' from the menu bar or by ***editing*** the ***input file***). The numbering of the sources (defined in the corresponding map file) is not necessarily sequential. The only demand, regarding the numbering of the sources, is that each source has a **unique ID**. To define a fixed point to be added to the grid but not to be regarded as a source-node requires some additional ***editing*** of the ***input file***.

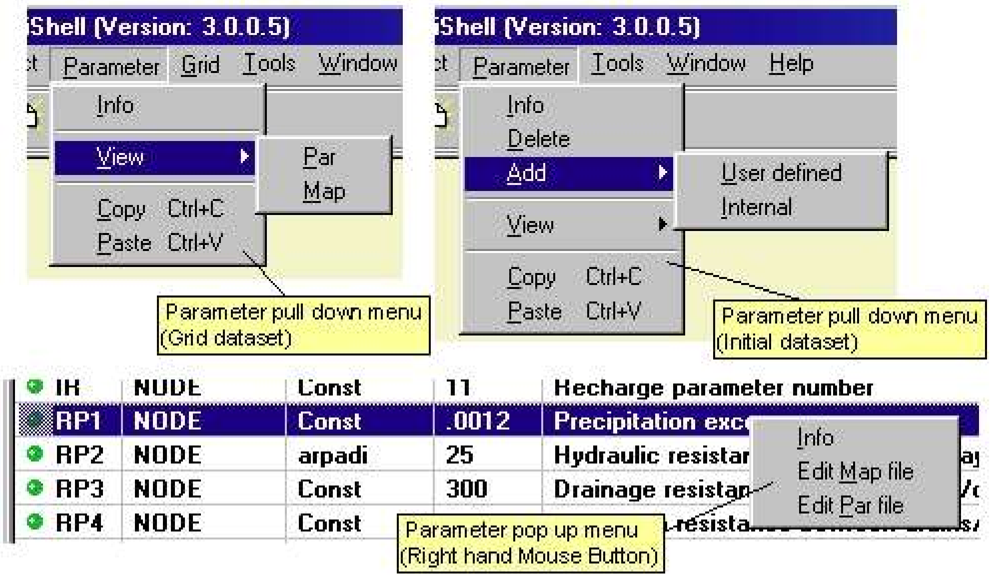
- The size of the elements within areas defined by the node distance of the ***density polygon*** map (the parameter **POL** of the ***grid data set***).

Summarizing, the four grid specific parameters which define the structure of the Finite Element/Difference Mesh are:

- **BND** : Map file containing one single polygon, defining the boundary of the model's domain
- **POL** : Map file containing a number of polygons, each defining an area with user defined node distances
- **RIV** : Map file containing a number of lines, each defining a river, channel or other waterway.
- **SRC** : Map file containing a number of points, each defining the location of a groundwater abstraction or infiltration

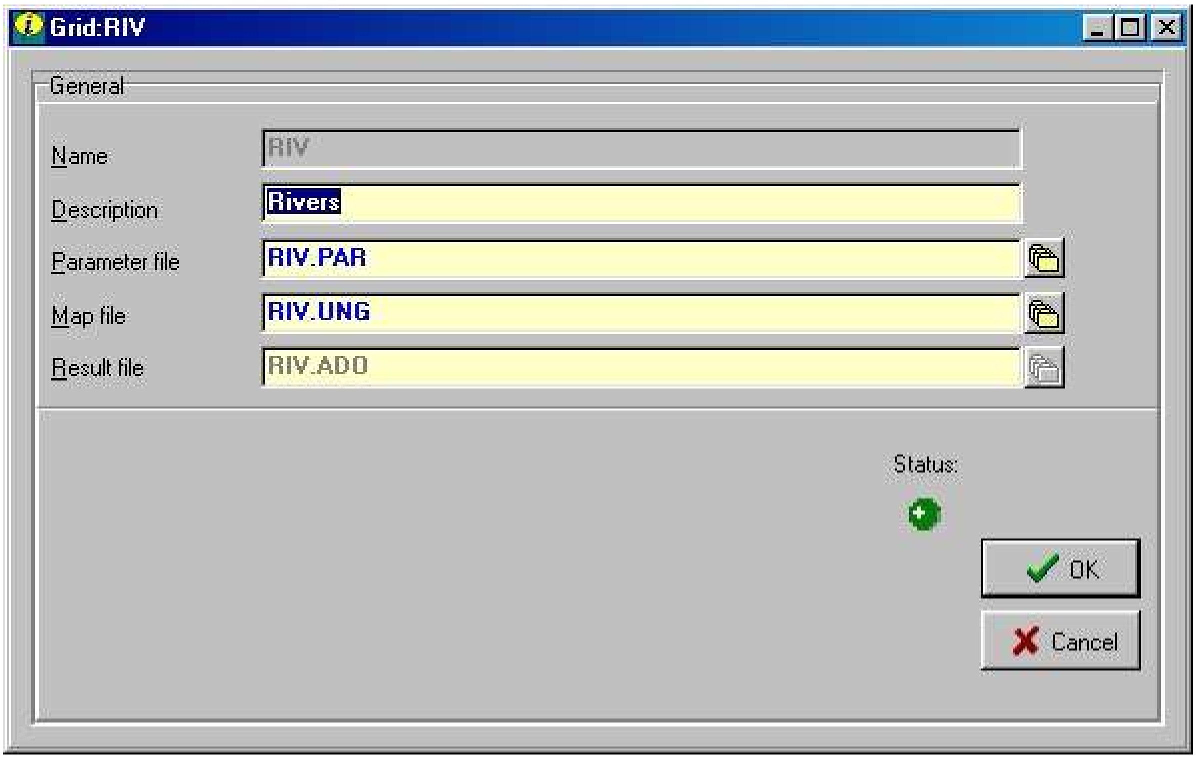


Double clicking on one of the parameters causes the graphical editor DigEdit (How to use DigEdit is explained in chapter 8) to open. For each of the grid parameters the user creates a ***map file*** containing the topographical layout of that parameter within the model's domain.

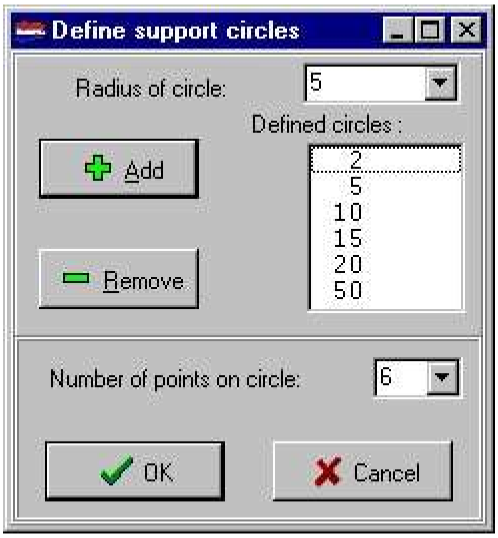


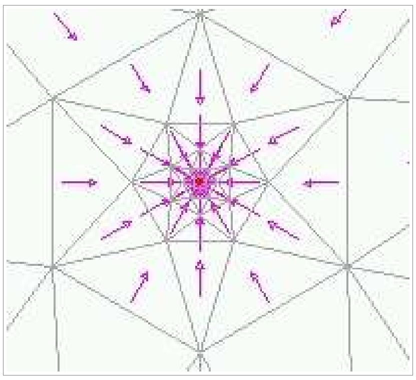
Pressing the right hand mouse button displays a pop-up menu which allows to retrieve 'Info' or to 'Edit' the map file or parameter values file (the ***par file***). Choosing 'Parameter' from the menu bar displays a pull down menu with a slightly more comprehensive selection of possibilities: 'Info', 'Delete', 'Add' ('User defined' or 'Internal'), 'View' ('Map' or 'Par'), 'Copy' and 'Paste'. Accessing the 'Parameter' pull down menu while the Grid data set is active the options 'Delete' and 'Add' are omitted because only the four parameters mentioned are used and all four are needed.

Selecting 'Info' from the pull down menu displays the parameter's name, the type of parameter selected, the names of the map, parameter and result files used to define the parameter and the status of the parameter. The status indicator shows whether or not map and par files have been defined and whether the parameters have been allocated or not.



The item 'Grid' has been added to the menu bar. Selecting 'Grid' displays the Grid pull down menu. This menu allows the user to generate the Grid and to view the results.





Selecting 'Define support circles' from the pull down menu allows the user to add one or more ***Support circles*** to the sources nodes. The user can choose from a number of predefined radii and sets the number of nodes to be generated on the support circles by selecting the appropriate items from the dialog window.

The Support circles allow the user to define a locally very dense grid, which improves the results of the calculation of groundwater flow in the vicinity of abstraction or infiltration wells. Because of the nature of the finite difference grid this option is available for finite elements only.

4.3.4 Generating the Grid

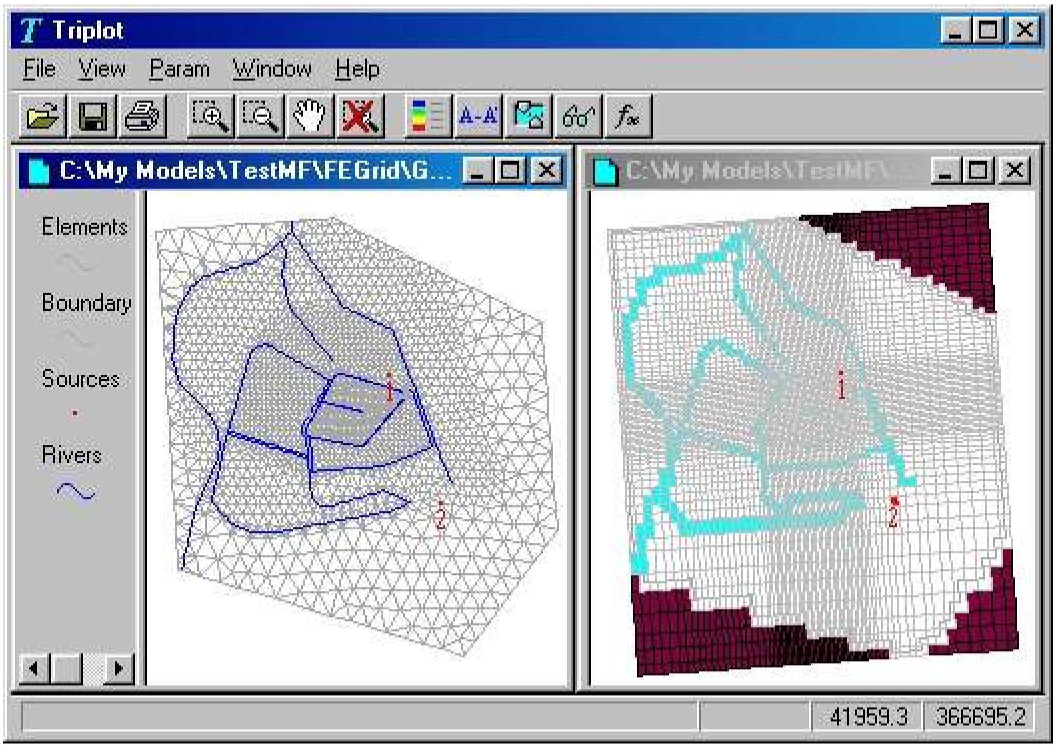
Once a map file is created for all grid parameters, the grid can be generated. Select 'Generate Input file' from the pull down menu to create the ***grid.tei*** input file needed for the grid generator. The input file may be viewed selecting 'View' 'Input', which opens the input file using the default text editor. See paragraph 4.3.6 for the input data description of the ***grid.tei***.

To start the grid generator one should select 'Start grid generation' from the pull down menu. TriShell starts the grid generator in a separate window.

Alternatively, the grid generate program may be run stand-alone choosing the corresponding icon from the **Triwaco** Program Folder. In that case, however, the program will not be displayed in the Tasks window.

The grid generator writes the results to the standard ASCII text file ***grid.teo***. This file can be viewed (in text mode) selecting 'View' 'Grid output as text' from the pull down menu. Selecting 'View' 'Print' opens the execution log file (***grid.tep***), which contains all information regarding the grid generating process. See paragraph 4.3.7 for an example output file.

4.3.5 Viewing the Grid



The resulting Grid may be viewed selecting 'View' 'Grid' from the Grid pull down menu. This starts the graphical presentation tool TriPlot (see also chapter 9) loads the grid information and displays the layout of the model's area.

4.3.6 Input data description

The input file (***grid.tei***) for generation of a Finite Element grid or Finit Difference grid is a readable ASCII text file. For the generation of a Finite Element grid the program **Tesnet** is used. For generating a Finite Difference grid the program **Monet**, is used for which, because of the nature of the Finite Difference grid, restrictions apply (paragraph 5.4.2). In some cases an alternative gridgenerator may be used (paragraph 4.3.8). The ***grid.tei*** contains the following contents:

Set 1:

HEAD	· identification of project or grid	Format A40

HEAD is an alphanumerical string for identification of the project's grid

Set 2:

NBP, NRIV, NSRC, NPOL, EPFIX	· number of boundary input points · number of rivers, line elements · number of sources, fixed points · number of density polygons · absolute minimum distance between fixed and nodal points	Format Free

NBP, NRIV, NSRC and NPOL are integer values and ≥ 0 (the value is obtained from the corresponding parameter map files BND, RIV, SRC and POL);
EPFIX is a real value ≥ 0. EPFIX may be defined by polygons to vary EPFIX. The file should be defined by the parameter name EPFIX.

Set 3a:

XB1, YB1	· coordinates first boundary point	Format Free

XB1, YB1 are real values

Set 3b:

XB*i*, YB*i*, IBP	· coordinates next input point (*i* = 2, …, NBP) · code for generation BND nodes	Format Free

XB*i*, YB*i* are real values; the coordinates of the last boundary point (XB_{NBP}, YB_{NBP}) should be equal to those of the first boundary point (XB1, YB1).
IBP is an integer, either -1 (default) or >0
. If IBP = -1 the number of nodes generated depends on the node density.
. If IBP > 0 the number of nodes generated between boundary point '*i*' and boundary point '*i*-1' equals IBP.

Set 3b will be repeated (NBP-1) times.

Set 4a:

XR1, YR1, IRIV, Nrivp	· coordinates first river point · river ID · total number river input points	Format Free

XR1, YR1 are real values
IRIV is an integer value ≥ 0;
. If IRIV=0 the line is not considered a 'river' and is not included in the number of rivers NRIV (Set 1). Lines with IRIV=0 should be preceded and followed by lines with IRIV≥0. More than one line with IRIV=0 may be present in the input file.
Nrivp is an integer value >2

Set 4b:

XR*i*, YR*i*, IRIV, IRP, WIDTH	· coordinates nest input point (*i* = 2, …, Nrivp) · river ID · code for generation river nodes	Format Free

XR*i*, YR*i* are real values
IRIV is an integer value ≥ 0; the same as for Set 4a.

IRP is an integer, either -999, -1 (default) or >0

. If IRP = -999	the number of nodes generated equals the node density.
. If IRP = -1	the number of nodes generated equals half the node density.
. If IRP > 0	the number of nodes generated between river input point '*i*' and river input point '*i*-1' equals IRP.

WIDTH is an optional real value ≥ 0.
. If WIDTH is given TESNET generates an additional line at both sides of the river defined by the coordinates, the distance of the additional lines to the central line being equal to WIDTH.

Set 4b will be repeated (Nrivp-1) times.

Set 4a and 4b will be repeated (NRIV + N_{R0}) times.
N_{R0} being the number of times a river has been defined with IRIV=0.

Set 5a:

XS, YS, Ncir, Npc	· coordinates of source point · nr of support circles · nr of nodes to be generated on each support circle	Format Free

XS, YS are real values
Ncir is an integer value, either -1, 0 (default) or >0;

. If Ncir = 0	there are no support circles and set 5b should be skipped.
. If Ncir = -1	there is only one support circle, the radius of the support circle will be read from an additional value R1 on the same record: XS, YS, Ncir, Npc, R1. Set 5b should be skipped.
. If Ncir > 0	Ncir support circles are present, the radii of which are given in set 5b.

Npc is an integer value, either -1, 0 (default) or >0.

. If Npc > 0	Npc points are added to each support circle.
. If Npc = -1	there is no support circle, the point is not considered a 'source' and is not included in the number of sources NSRC (Set 1). Lines with Npc=-1 should be preceded and followed by sets defining regular sources. More than one point with Npc=-1 may be present in the input file.

Set 5b:

R1, R2, …, R*i*, …, Rncir	· radii of support circles	Format Free

R1, R2, R*i* etc are real values ≥ 0.

Set 5b should be skipped if Ncir equals -1 or 0.

Set 5a and 5b will be repeated (NSRC + N_{FP}) times.
N_{FP} being the number of times a fixed point has been defined that is not a source; Npc=-1.

Set 6a:

IPOL, Npp, DIST, EPPOL	· sequential polygon number · nr of polygon input points · node distance for nodes generated within the polygon · minimum distance to previously generated nodes	Format Free

IPOL is an integer value >0
Npp is an integer value >3
DIST and EPPOL are real values >0, by default EPPOL equals half the value of DIST.

Set 6b:

XP*i*, YP*i*	· coordinate of polygon input point (*i* = 1, …, Npp)	Format Free

XP*i* and YP*i* are real values.

Set 6b will be repeated Npp times. The coordinates of the last input point (XP***Npp***, YP***Npp***) should be equal to the coordinates of the first input point (XP***1***, YP***1***).

Set 6a and 6b will be repeated NPOL times (Set 1).
The last polygon, having the largest node-distance, should cover the whole model area. Hence, all corner points of this polygon should be outside the model's boundary (parameter BND).

Example of a grid input file *grid.tei*

SET	Example text	Parameter	Description
1	FE Grid	HEAD	identification of project or grid
2	8 3 3 3 5	IBP, .. NRIV, .. NSRC, .. NPOL, .. EPFIX	number of boundary input points number of rivers, line elements number of sources, fixed points number of density polygons absolute minimum node distance
3 a	147006 410600	XB1, YB1	coordinates first boundary point
3 b	146783 409502 –1	XB2, YB2, IBP	
3 b	147224 408645 –1	XB3, YB3, IBP	
3 b	148316 407983 –1	.	IBP < 0; automatic generation
3 b	150005 408537 –1	.	->distance equal to node distance
3 b	149967 410841 25	.	IBP > 0; nr of nodes to generate
3 b	148472 411106 –1	.	-> distance = section length / nr intervals
3 b	147006 410600 –1	XBN, YBN, IBP	N = IBP (see set 2)
4 a	148844 411097 1 24	XR1, YR1, IRIV, Nrivp	coordinates first river point river ID total number of river points
4 b	148800 410974 1 –1	XR2, YR2, IRIV, IRP	coordinates next river point river ID code for generation RIV nodes
4 b	148682 410794 1 –1	XR3, YR3, IRIV, IRP	(default value for IRP = -1)
4 b	148550 410653 1 –1	.	IRP < 0; automatic generation
4 b	148494 410553 1 –1	.	Distance equal to 0.5 node distance
4 b	148479 410444 1 –1	.	
4 b	148488 410373 1 –999	.	IRP = -999; automatic generation Distance
4 b	148588 410405 1 -1	.	equal to node distance
4 b	148668 410179 1 -1	.	
4 b	148582 410014 1 -1	.	
4 b	148574 409899 1 -1	.	IRP < 0; automatic generation
4 b	148635 409687 1 11	.	IRP > 0; nr of nodes to generate
4 b	148706 409578 1 -1	.	
4 b	148759 409299 1 -1	.	
4 b	148862 409025 1 -1	.	
4 b	148956 408878 1 -1	.	
4 b	149118 408807 1 -1	.	
4 b	149250 408716 1 -1	.	
4 b	149283 408542 1 -1	.	
4 b	149289 408404 1 -1	.	
4 b	149274 408263 1 -1	.	
4 b	149289 408201 1 -1	.	
4 b	149359 408122 1 -1	.	
4 b	149445 408022 1 -1	XRN,YRN,IRIV,IRP	N = Nrivp (see set 4a)
4 a	148479 410373 2 15	XR1,YR1,IRIV,Nrivp	Repeat set 4 for next river
4 b	148285 410373 2 -1	XR2,YR2,IRIV,IRP	
4 b		.	
4 b		XRN,YRN,IRIV,IRP	
5 a	148351 409687 2 6	XS, YS, Ncir, Npc	coordinates source point nr of support circles (default value Ncir = 0) nr of nodes on support circle (default Npc = 0)
5 b	10 25	R1, R2	Radii of support circles nr of radii equals Ncir (set 5a)
			Repeat set 5a and 5b for each source point
6 a	1 9 10 3.3	IPOL,Npp,DIST,EPPOL	polygon ID number of points of polygon node distance minimum distance factor for polygon
6 b	148261 409843	XP1, YP1	coordinates first polygon point
6 b	148006 409675	XP2, YP2	coordinates next polygon point

SET	Example text	Parameter	Description
6 b	148010 409490	.	
6 b	148335 409308	.	
6 b	148726 409420	.	
6 b	148698 409567	.	
6 b	148628 409675	.	
6 b	148568 409881	.	
6 b	148261 409843	XPN, YPN	N = Npp (see set 6a)
6 a	2 10 50 16.5	IPOL,Npp,DIST,EPPOL	Repeat set 6 for next polygon
6 b	148469 410459	XP1, YP1	
6 b			
6 b	148469 410459	XPN, YPN	
6 a	3 8 250 82.5	IPOL,Npp,DIST,EPPOL	Repeat set 6 for next polygon
6 b	146916 410810	XP1, YP1	
6 b	146518 409435	.	Nr of sets equals NPOL (set 2)
6 b	146991 408430	.	
6 b	148311 407923	.	
6 b	150093 408445	.	
6 b	150202 410885	.	
6 b	148490 411174	.	
6 b	146916 410810	XPN, YPN	
File ends with an empty line			

END OF FILE

4.3.7 Output data description

The grid generation program creates a formatted sequential file containing all information about the Finite Element or Finite Difference grid generated. The output file (***grid.teo***) consists of 8 data records and 13 parameter arrays or adore-sets, the standard **Triwaco** format.

Note The information for the Finite Difference is also saved in the ***grid.teo*** file. Upon execution of the ModFlow simulation the grid and paramater data is converted to standard ModFlow format. In addition the grid information is also saved as Finite Element data.

The first record contains the project identification. The next seven records (2 through 8) contain info concerning the finite element grid. This information consists of literal text followed by an integer number:

Description	Type	Tesnet, Trinet	Monet	ReGuGrid
NUMBER NODES	NOD	X	X	X
NUMBER ELEMENTS	NEL	X	X	X
NUMBER FIXED POINTS	NFIX	X	X	
NUMBER SOURCES	NSRC	X	X	
NUMBER RIVERS	NRIV	X	X	
NUMBER RIVER NODES	NRP	X	X	
NUMBER BOUNDARY NODES	NBP	X	X	X
NUMBER OF ROWS			X	X
NUMBER OF COLUMNS			X	X
NUMBER SOURCE CELLS			X	
NUMBER RIVER CELLS			X	
ROTATION ANGLE			X	

Depending on the type of grid generator used adore sets with the following labels are written to the grid output file:

	Finite Element
1	X-COORDINATES NODES
2	Y-COORDINATES NODES
3	ELEMENT NODES 1
4	ELEMENT NODES 2
5	ELEMENT NODES 3
6	ELEMENT AREA
7	NODE INFLUENCE AREA
8	SOURCE NODES
9	NUMBER NODES/RIVER
10	LIST RIVER NODES
11	LIST BOUNDARY NODES
12	BOUNDARY SEGMENTS
13	RIVERNUMBER
14	SOURCENUMBER
	Finite Difference
16	DELC
17	DELR
18	INACTIVE CELLS
19	SOURCE CELLS
20	RIVER CELLS
21	RIVER LENGTH

The parameter names of theadore-sets are self-explanatory. Sets 8 and 14 (SOURCE NODES, SOURCENUMBER) are omitted if the number of sources equals 0. Sets 9, 10 and 13 (NUMBER NODES/RIVER, LIST RIVER NODES, and RIVERNUMBER) are omitted if the number of rivers equals 0.

Furthermore, a second output file is generated with the default name ***grid.tep***. This ***print output file*** consists of an echo of the input, some intermediate results, and data of the generated grid. The ***print output file*** is useful to track a possible error in the input file. The file contains the number of boundary nodes, river nodes and source nodes that have been read and generated by the program. Moreover, nodes that are eliminated or moved because their distance to neighboring points is less than the specified minimum distance (EPFIX) are listed and the remaining number of boundary, river and source nodes is printed. Once the grid has been generated, the minimum and maximum element area and the coordinates of the nodes are printed.

4.3.8 Alternative grid generators

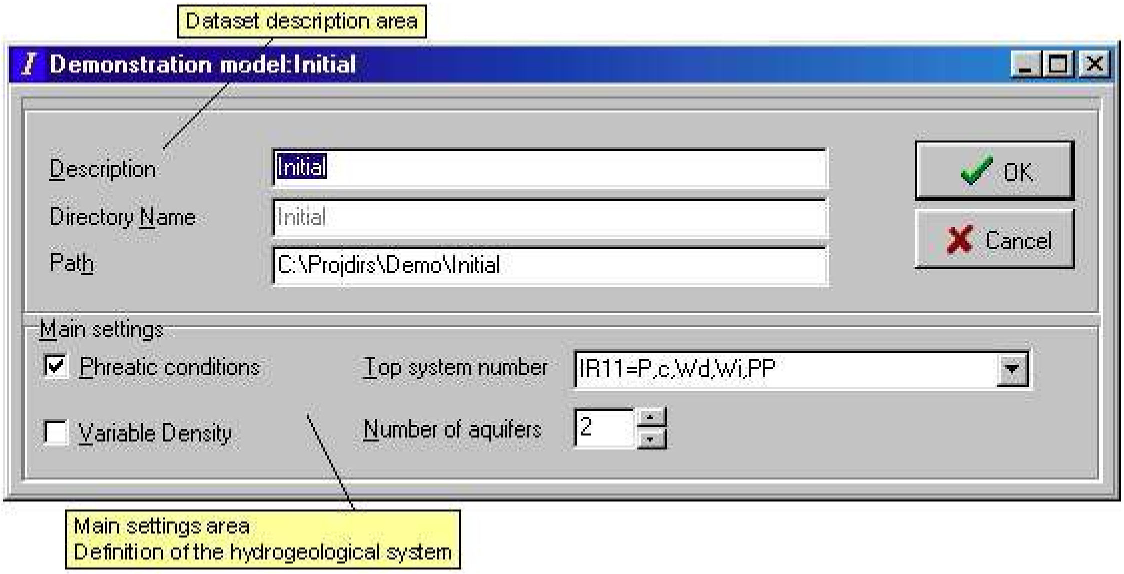
Trinet
In addition to the standard grid generation program **Tesnet** and **Monet** there are other grid generation programs included. One of these is **Trinet**, which is a Finite Element grid generator (TIN). The program is much faster but has some restrictions. **Trinet** does not support the generation of support-circles around sources or the generation of rivers consisting of multiple parallel line elements. It reads the standard grid.teo input file and generates a standard **Triwaco** grid output file.

ReGuGrid
In addition to the standard grid generation program **Tesnet** and **Monet** there are other grid generation programs included. One of these is **ReGuGrid**, which produces a Finite Difference grid. The cells of a grid generated with ReGuGrid are all equally sized (equal width and height). This grid has the same restrictions as as grids generated by Monet. Additionally sources, rivers and density polygons are ignored. For the definition of the cell size ReGugrid uses the smallest value from the density polygons (if more than one is defined).

4.4 Creating an Initial data set

4.4.1 Introduction
In the initial data set the user defines the conceptual model. All original data is stored grid independently, so it is possible to make a model with different (type of) grids but with the same initial data.

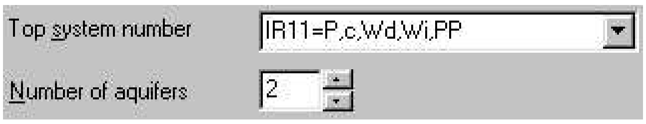
4.4.2 Opening an Initial data set
Selecting 'data set' 'Add' from the pull down menu and 'Initial' from the ***'create new data set'*** dialog window the ***'initial data info window'*** is displayed and the user has to provide information regarding the hydrogeological system. The ***'initial data info window'*** is divided in two parts. In the upper part a description, the directory name and the path have to be given. In the lower part of the window the properties defining the hydrogeological system are recorded.



4.4.3 Defining model properties
The properties defining the hydrogeological system are recorded in the ***'Main settings'*** area, the lower part of the ***'initial data info window'***. Subsequently the following information has to be provided by checking the tick box ( ) or leaving it blank ( ).

Description	Unchecked box ( )	Checked box ( )
Phreatic conditions	All aquifers are **confined**	**Phreatic** upper aquifer
Variable Density*	**No Variable Density** is used	**Variable Density** is used
Transient	**Steady state** calculations	**Transient** calculations
Unsaturated zone modeling	**No modeling** of vertical groundwater flow in the **unsaturated zone** (FLUZO)	**Modeling** of vertical groundwater flow in the **unsaturated zone.**

* When a model is created for using the Variable Density additional parameters have to be defined (see also chapter 14).



At the right hand side of the ***'Main settings'*** area the number of aquifers can be selected in the corresponding box and the type of topsystem, representing the upper boundary condition, can be selected.

Topsystems
The discharge or recharge of groundwater at the top of the first aquifer can be characterized by the so-called ***top-systems***. A top-system describes the interaction between the groundwater system and a drainage/infiltration system consisting of generally small surface waters and drains. A short description of the topsystems is listed below. A more detailed description is given in Appendix A.

1. Precipitation; Top-system number 1, defined by 1 parameter; groundwater recharge is equal to the precipitation excess.
2. Polder with fixed water level; Top-system number 2, defined by 3 parameters; groundwater recharge and discharge depend on a fixed water level and the (total) resistance of the drainage/infiltration system.
3. Phreatic drainage; Top-system number 3, defined by 3 parameters; groundwater discharge depends on the head in the top aquifer, the resistance and the base of the drainage system.
4. Three-level drainage system; Top-system number 4, defined by 13 parameters; groundwater recharge or discharge depends on the precipitation excess and the resistance and levels of a primary, secondary and tertiary drainage/infiltration system.
5. Pipe drainage and irrigation or precipitation; Top-system number 5 (drainage only) and Top-system number 6 (both drainage and infiltration), defined by 8 parameters; groundwater discharge depends on the precipitation or irrigation excess, the head in the top aquifer and the drainage resistance.
6. Polder with a fixed water level and precipitation; Top-system number 7, defined by 4 parameters; groundwater recharge or discharge depends on a fixed water level, the (total) resistance of the drainage system and the precipitation excess.
7. Phreatic drainage with precipitation; Top-system number 10, defined by 4 parameters; groundwater discharge depends on the head in the top aquifer, the resistance and the base of the drainage system and on the precipitation excess.
8. Polder with a fixed water level and single drainage system; Top-system number 11, defined by 5 parameters; groundwater recharge or discharge depends on the precipitation excess and the resistance and level of a single drainage system.
9. Predefined recharge or discharge characteristic; Top-system number 12, defined by 5 parameters; groundwater recharge or discharge depends on meteorological quantities and soil parameters. The soil parameters are obtained by curve fitting of the Van Genuchten relations.

IR	RP1	RP2	RP3	RP4	RP5	RP6	RP7	RP8	RP9	RP10	RP11	RP12	RP13
1	P												
2	H_P	C_0	W										
3	H_S	W	BD										
4	P	C_0	H_P	$W_{d,1}$	$W_{d,2}$	$W_{d,3}$	$W_{i,1}$	$W_{i,2}$	$W_{i,3}$	BD_1	BD_2	BD_3	H_S
5	P	H_S	H_d	H_T	K_v	K_h	L	R					
6	P	H_S	H_d	H_T	K_v	K_h	L	R					
7	P	C_0	W	H_P									
8	**not in use**												
9	**not in use**												
10	P	W	BD	H_S									
11	P	C_0	W_d	W_i	H_p								
12	P	ET_{mx}	a	b	H_S								

As can be noticed from this table the top system parameters **RP*xx*** for different top systems do not necessarily represent the same physical parameter. For example, parameter RP1 may represent precipitation (*P*), the surface level elevation (*Hs*) or the controlled water level (*Hp*). Moreover, different top systems require a different number of parameters, ranging from only one (for top system type 1) to as much as thirteen (top system type 4).

The physical parameters associated with the top system parameters are listed in the following table. One can distinguish parameters related to the meteorological condition (precipitation and evapotranspiration), soil parameters, surface and surface water levels and parameters with respect to the geometry and resistances of the drainage system.

Selecting a top system from the list with predefined sets causes the program to load the corresponding number of top system or recharge parameters. Similarly, the program also loads the appropriate number of aquifer parameters, taking into account the number of aquifers specified and the type of aquifer condition for the upper and lowermost aquifer.

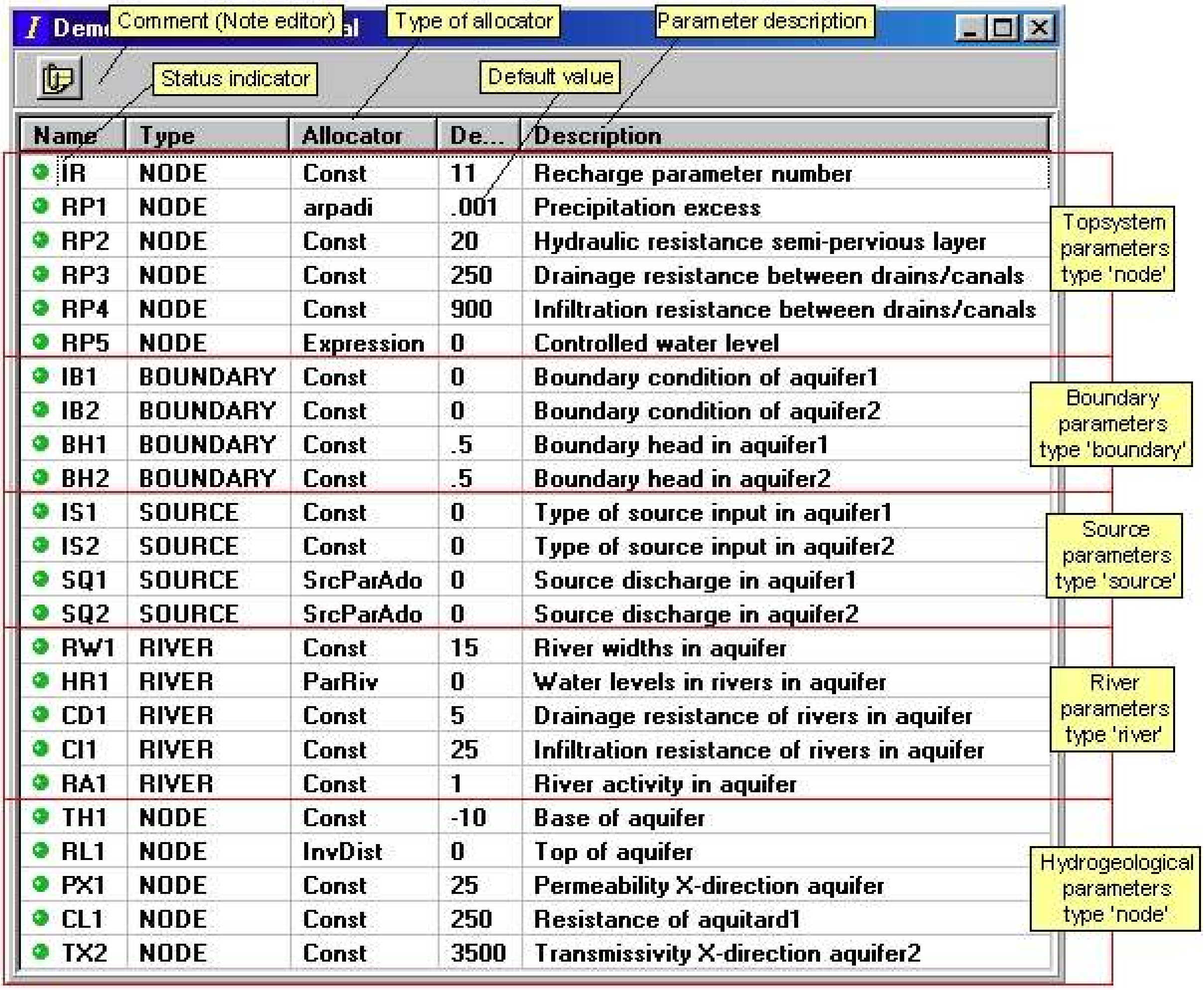
Confirming the selection with the 

-button causes the program to open the ***'Initial data set window'***, displaying all model parameters needed. For each of the model parameters a ***map*** and ***par file*** may be defined or the parameter may be assigned a constant value.

Name	Definition of parameter
P	Precipitation excess or irrigation excess
ET_{mx}	Maximum Evapotranspiration
A	soil parameter obtained by curve fitting
B	soil parameter obtained by curve fitting (b > 1)
C_0	Hydraulic resistance of semi-pervious top layer
H_d	Drain level of system of (pipe)—drains
H_P	Polder water level or controlled water level
H_S	Surface level (with respect to the ordnance level)
H_T	Level of base of semi-pervious top layer
K_h	Horizontal permeability of semi-pervious top layer
K_v	Vertical permeability of semi-pervious top layer
L	Horizontal distance between drains
R	Wetted perimeter of (pipe)—drains
BD	Drainage base or bottom level of the (open) drains
BD_1	Drainage base or bottom level of the primary drainage system
BD_2	Drainage base or bottom level of the secondary drainage system
BD_3	Drainage base or bottom level of the tertiary drainage system
W	Drainage or infiltration resistance between ditches or drains
W_d	Drainage resistance between ditches or drains
$W_{d,1}$	Drainage resistance of the primary drainage system
$W_{d,2}$	Drainage resistance of the secondary drainage system
$W_{d,3}$	Drainage resistance of the tertiary drainage system
W_i	Infiltration resistance between ditches or drains
$W_{i,1}$	Infiltration resistance of the primary drainage system
$W_{i,2}$	Infiltration resistance of the secondary drainage system
$W_{i,3}$	Infiltration resistance of the tertiary drainage system

4.4.4 Defining model parameters (general)

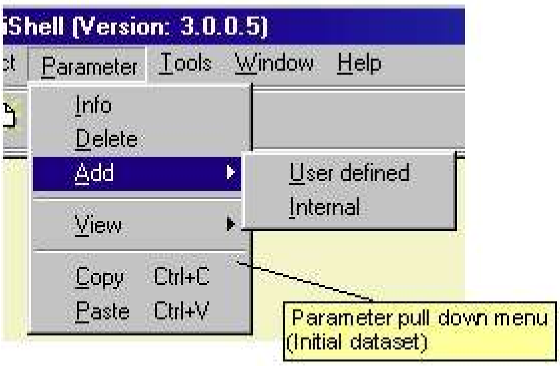
To define a model parameter the user has to provide a ***map*** and ***par file*** and has to specify the allocator (appenix C) to be used. The allocator defines how parameter values are assigned to the nodes of the grid. **Triwaco** opens the ***'Initial data set window'*** with a set of default allocators, depending on the type of parameter. An overview of parameter types is listed here (see appendix B for a complete overview and the lay out of the map, parameter and corresponding ado files).

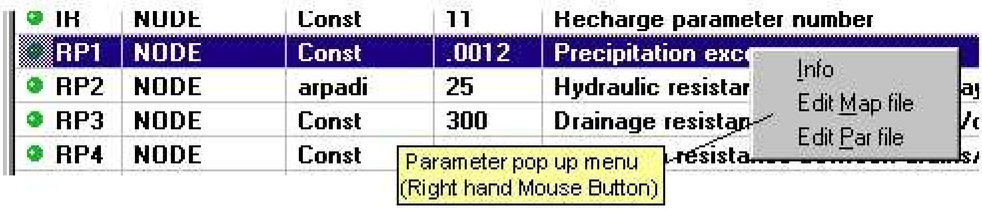


Name	Type	Allocator	De...	Description
IR	NODE	Const	11	Recharge parameter number
RP1	NODE	arpadi	.001	Precipitation excess
RP2	NODE	Const	20	Hydraulic resistance semi-pervious layer
RP3	NODE	Const	250	Drainage resistance between drains/canals
RP4	NODE	Const	900	Infiltration resistance between drains/canals
RP5	NODE	Expression	0	Controlled water level
IB1	BOUNDARY	Const	0	Boundary condition of aquifer1
IB2	BOUNDARY	Const	0	Boundary condition of aquifer2
BH1	BOUNDARY	Const	.5	Boundary head in aquifer1
BH2	BOUNDARY	Const	.5	Boundary head in aquifer2
IS1	SOURCE	Const	0	Type of source input in aquifer1
IS2	SOURCE	Const	0	Type of source input in aquifer2
SQ1	SOURCE	SrcParAdo	0	Source discharge in aquifer1
SQ2	SOURCE	SrcParAdo	0	Source discharge in aquifer2
RW1	RIVER	Const	15	River widths in aquifer
HR1	RIVER	ParRiv	0	Water levels in rivers in aquifer
CD1	RIVER	Const	5	Drainage resistance of rivers in aquifer
CI1	RIVER	Const	25	Infiltration resistance of rivers in aquifer
RA1	RIVER	Const	1	River activity in aquifer
TH1	NODE	Const	-10	Base of aquifer
RL1	NODE	InvDist	0	Top of aquifer
PX1	NODE	Const	25	Permeability X-direction aquifer
CL1	NODE	Const	250	Resistance of aquitard1
TX2	NODE	Const	3500	Transmissivity X-direction aquifer2

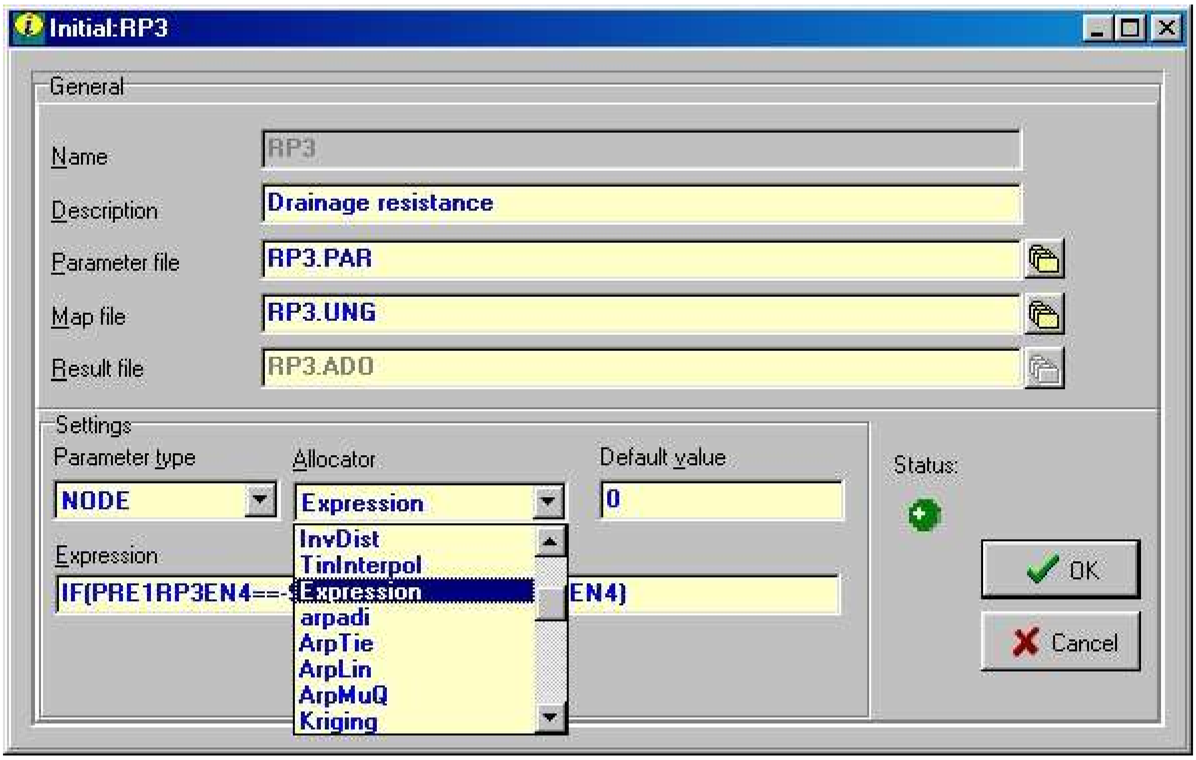
Double clicking on one of the parameters starts the graphical editor DigEdit (Chapter 8). If map and par files exist for the parameter considered these files are loaded, if not the screen remains empty. For each of the parameters the user creates a ***map file***. This file contains the topographical layout of the parameter concerned, consisting of a set of points, lines or polygons that are (partly) within the model's domain. Each graphical object in the ***map file*** will be assigned a value; these parameter values are stored in the ***par file***, containing the object's ID and the parameter value.

Pressing the right hand mouse button displays a pop-up menu which allows to retrieve 'Info' or to 'Edit' the map or par file. Choosing the 'Parameter' pull down menu from the menu bar displays a slightly more comprehensive selection of possibilities: 'Info', 'Delete', 'Add' ('User defined' or 'Internal'), 'View' ('Map' or 'Par'), 'Copy' and 'Paste'.





Selecting 'Info' from the pull down menu displays the '***parameter info window***', with the parameter's name, the type of parameter selected, the names of the map and par files used to define the parameter and the status of the parameter. The status indicator shows whether or not the map and par files have been defined and whether the parameter has been allocated or not. The '***Settings area***' of the '***parameter info window***' allows the user to change the parameter type, the allocator and the default value. Moreover it allows the definition of an expression, which relates the selected parameter to other model parameters.



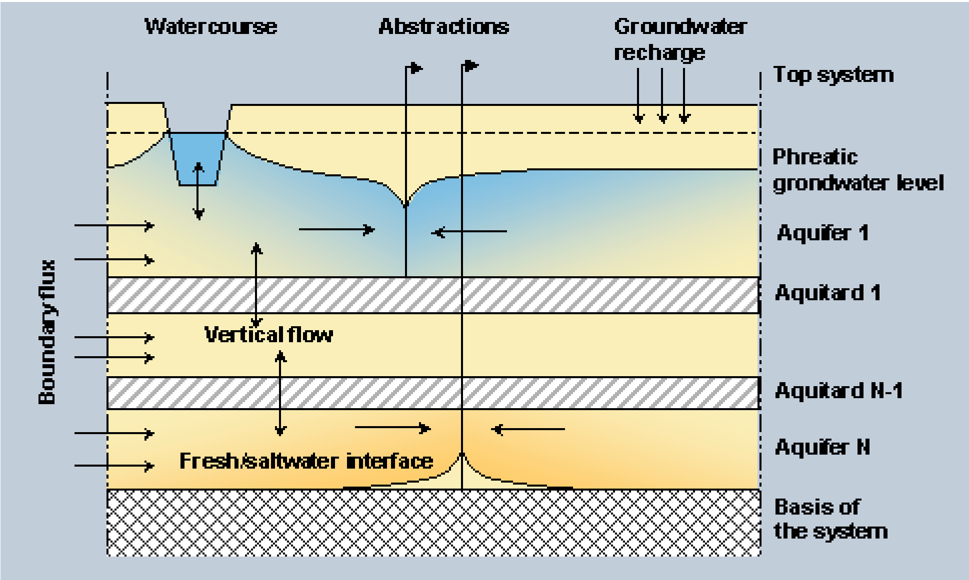
The name in the ***General*** information area is the predefined parameter name that is recognized by **Triwaco**. The description may be modified; this is a short descriptive comment characterizing the parameter. The names of the parameter, map and result file are generally the same as the parameter name and differ only by their extension.

In the ***Settings*** area the proper ***allocator type*** has to be provided and the default value for the parameter considered has to be given. This deafault value will be assigned to the parameter if the allocator type is set to "Const" or for parts of the model's domain that are not covered by the parameter's map file.

Triwaco includes a range of powerful geo-processors for 1D to 4D interpolation. The processors are called allocators since they are used to assign (allocate) parameter GIS maps/values to the individual nodes or cells of the grid. Most allocators can be used for different types of parameters. For source, river and boundary parameters specific allocators are available. Other allocators are used for distributed parameters only (assigning a parameter value to each node of the grid). In appendix C descriptions and usage of all allocators can be found.

Optionally the parameter may be related to other parameters by a (mathematical) expression. The allocator type has to be set to "Expression" and the expression itself should be entered in the Expression-box (see appendix C for all options using the expression allocator).

After having provided all information needed, including the necessary map and par files, the status indicator of the parameter changes from to .



4.4.5 Definition of boundary conditions

The type of boundary condition is defined by the parameter IB***i***. Multiple type boundary conditions may be defined for different parts of the model boundary. For those parts of the model boundary for which IB***i***=0 a constant head boundary applies (default). A constant head boundary implies the definition of the boundary head by the parameter BH***i***, which defines the constant head in aquifer ***i***.

For IB***i***=1 a constant flux boundary applies. Consequently the constant flux has to be defined by the parameter BB***i***, which defines a constant flux (m3/d per m) in aquifer ***i***, and/or the parameter BA***i***, which defines the the slope of the boundary flux (m3/d per m2) in aquifer ***i***. The flux is defined as Q=BA * PHI + BB, where PHI is the groundwater head on the boundary.

Boundary type parameters are defined by so called `Linked Points` in DigEdit (chapter 8). A linked point is used to assign values to grid parameters: boundary and rivers(line elements). These points, when used to define boundary conditions, by definition are linked to ID 1, which represents the ID of the boundary. Each point is given a value for flux or head depending on the condition defined. These parameters are allocated with the allocator ParBou. This allocator will interpolate (lineair) between the points.

4.4.6 Definition of river (line-element) parameters

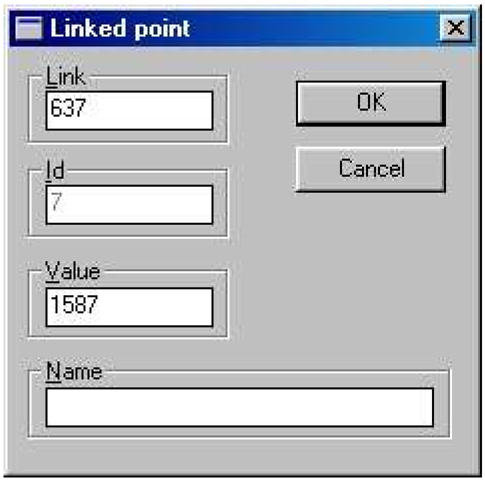
The river activity is controlled by the parameter RA***i***.

RA=0

The line elements for which RA*i*=0 are inactive and are treated as regular nodes/cells during the simulation.

RA=1
A line element or river for which a constant head applies RA*i*=1. The properties of the line element or river are defined by four parameters; HR***i*** defines the waterlevel or head in aquifer ***i***, RW***i*** defines the width in aquifer ***i***, CD***i*** defines the drainage resistance in aquifer ***i***, CI***i*** defines the infiltration resistance in aquifer ***i.***

RA=2
A line element or river for which a constant discharge/recharge applies Ra*i*=2. A HOrizontal BOring (HOBO) or a range of small wells can be schematised as a single or multiple line-element in aquifer ***i***. A HOBO is a line element (river) representing the wells in a section. For each line element an abstraction rate can be defined. The model will calculate the water level for that particular section at the given abstraction rate.



The properties of the line-element are defined by five parameters; HR***i*** defines the initial waterlevel or head in aquifer ***i*** , RQ***i*** defines the discharge in aquifer ***i***, RW***i*** defines the width in aquifer ***i***, CD***i*** defines the drainage resistance in aquifer ***i***, CI***i*** defines the infiltration resistance in aquifer ***i.*** HR is the initial waterlevel defined by the user and should be close to the expected waterlevel for iteration purposes.

In addition line-elements can also be clustered. The discharge, defined by RQ, of individual line-elements are evenly distributed in such a way that the head (or waterlevel) of all clustered line-elements will be the same.

The line-elements to be clustered are linked to the main line-element by the parameter RC***i,*** which contains linked points. For each line-element a linked point is used to link it to the main line-element, as shown in the screenshot. In this case line-element with ID 637 is linked to the main line-element with ID 1587.

RA=3
A line element or river for which a constant head applies RA*i*=3. This option equal to RA*i*=1 until PHI1 drops below the bottom of the river (BR*i*). In that case the flux is no longer governed by (HR-PHI)/CI but is limited to a maximum flux governed by (HR-BR)/CI. Which corresponds to way the fluxes are governed by the topsystem.

The properties of the line element or river are defined by five parameters; HR***i*** defines the waterlevel or head in aquifer ***i***, RW***i*** defines the width in aquifer ***i***, CD***i*** defines the drainage resistance in aquifer ***i***, CI***i*** defines the infiltration resistance in aquifer ***i***, BR***i*** defines the bottom level of the river in aquifer ***i.***

4.4.7 Definition of source parameters

The source activity is controlled by the parameter IS*i*.

IS=0
Sources for which a constant abstraction/injection rate applies IS*i*=0. The abstraction(-) and injection(+) are defined by the parameter SQ***i*** in aquifer ***i***.

IS=1
Sources for which a constant head applies IS*i*=1. The head is defined by the parameter SH***i*** in aquifer ***i***.

IS=2
In addition sources with a constant abstraction/injection rate may be clustered. This option is usefull when modelling a section of wells with one discharge point (suction pipe). Then for this point the abstraction rate can be defined (in fact the sum of all is taken). The model will calculate the water level for that particular section at the given abstraction rate.
Another case may be a source with multiple well screens each in a different aquifer. Then for this source the abstraction rate can be defined. The model will calculate the water level, which will be the same in all well

screens, at the given abstraction rate.

Sources are clustered by defining the parameter SN***i***. Sources for which SN=0 are not clustered. Sources for which SN=n are clustered. So when SN=22 these sources are part of cluster 22, etc.

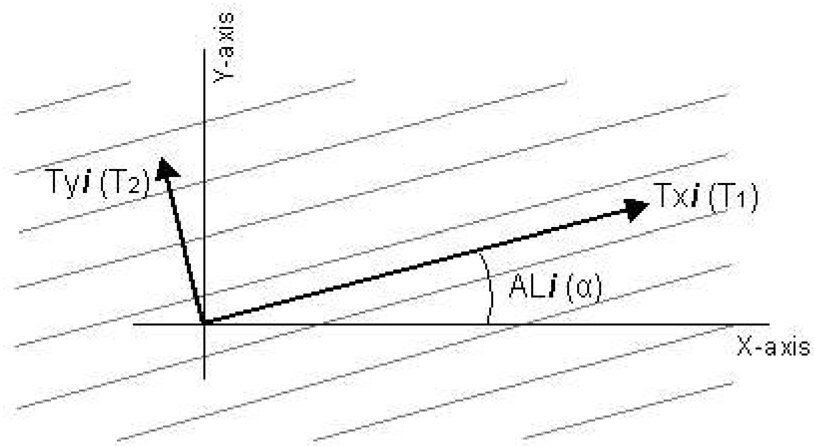
4.4.8 Definition of hydrogeological parameters

For confined conditions the transmissivity in each aquifer is generally defined by TX***i*** (m2/d). **Triwaco** does recognize permeability (PX***i***) provided top and base of the aquifer, respectively RL***i*** and TH***i*** , is defined as well. For phreatic conditions in the upper aquifer (aquifer 1) **Triwaco** calculates the transmissivity based on the permeability (PX1), base of the aquifer (TH1) and the calculated grondwatertable (PHIT). The top of the aquifer (RL1) needs also to be defined to account for situations wth groundwatertables rising above groundlevel. The resistance of each aquitard is defined by CL***i*** (d).

4.4.9 Definition of anisotropy

Although **Triwaco** assumes that transmissivities and permeabilities of all aquifers are by default isotropic, the user can define an anisotropic transmissivity (or permeability). For **ModFlow** the transmissivity tensor can only be defined in the K_x, K_y and K_z direction co-linear to the grid. Whereas for **Triwaco-Flairs** the transmissivity (or permeability) tensor may vary through the model area, which implies that the principal axes of the tensor can have different orientations in different points of the model domain (K_{xx}, K_{xy}, K_{yx}, K_{yy}, K_{zz}). So when anisotropy is important **Triwaco-Flairs** is the prefered simulation program. The input description therefore concentrates on **Triwaco-Flairs**.

For confined conditions the transmissivity in each aquifer is defined in the direction of the principal axis by TX***i*** and TY***i***. **Triwaco** does recognize permeability (PX***i*** and PY***i***) provided top and base of the aquifer, respectively RL***i*** and TH***i*** , is defined as well. The angel between the direction of TX and the positive X-axis is defined by AL***i***.



Parameter type	Preferred allocator	Parameter name
Type of Boundary condition	ParBou	IB***i***
Type of Source Input	SrcParAdo	IS***i***
River Activity	ParRiv	RA***i***
Type of Top system	Const	IR
Boundary Condition parameter	ParBou	BH***i***, BA***i***, BB***i***
Source parameters	SrcParAdo	SQ***i***, SH***i***, SN***i***
River parameters	ParRiv	HR***i***, RW***i***, CD***i***, CI***i***, ***RQi***
Distributed parameters	Various available	RP***xx***, CL***i***, TX***i***, PX***i***, etc.

4.4.10 Definition of expressions

General

The Expression allocator evaluates an expression and calculates (creates) a new ***Adore***-block. An expression may contain set-names, numbers, functions, factors and operators. Three types of operators may be distinguished: mathematical operators, relational operators and logical operators.

Definition	Description	
Set-names	Parameter names as defined in **Triwaco**, consisting of a combination of alphanumeric characters. The parameter may be preceded by the name of one of the project's data sets and a $-sign: e.g., cal$TX1	
Numbers	integer and real numbers: e.g., 15, -0.456	
Factors	Consist of numbers, expressions, functions or identifiers.	
Mathematical operators	+, -, * and /	
Relational operators	>, ≥ **(>=)**, = **(==)**, ≤ **(<=)** and <	
Logical operators	'AND' ('**&&**'), 'OR' ('||') and 'NOT' ('**=!**') and 'IF' 'THEN' ('**?**') and 'ELSE' ('**:**')	
Functions	(simple) mathematical functions:	
	abs(x)	Returns the absolute value of '*x*'
	atan(y,x)	Returns the arc tangent of ('*y/x*')
	BND(x)	Returns the value of '*x*' at boundary nodes
	cos(x)	Returns the cosine of '*x*'
	deg(x)	Converts radians ('*x*') to degrees
	exp(x)	Returns the value of **e** raised to the power '*x*'
	IF(x,y,z)	Evaluates the logical expression: IF ('*x*') THEN ('*y*') ELSE ('*z*') Equivalent to the expression: ('*x*')?('*y*'):('*z*')
	ln(x)	Returns the natural logarithm of '*x*'
	log(x)	Returns the 10 log of '*x*'
	max(x,y)	Returns the largest value of '*x*' and '*y*'
	min(x,y)	Returns the smallest value of '*x*' and '*y*'
	NODE(x)	Returns the value of '*x*' at **all Nodes**; if the value of '*x*' does not exist at a Node a zero value (**0**) is assumed
	rad(x)	Converts degrees ('*x*') to radians
	RIV(x)	Returns the value of '*x*' at river nodes
	sign(x)	Returns the sign of '*x*' (-1, 0 or +1)
	sin(x)	Returns the sine of '*x*'
	sqr(x)	Returns the square of '*x*'
	sqrt(x)	Returns the square root of '*x*'
	SRC(x)	Returns the value of '*x*' at source nodes
	tan(x)	Returns the tangent of '*x*'

Important note: The setname or data set name should NOT contain an underscore (data_set$set_name).

Examples of expressions

In the following table examples of the more or less frequently used expressions are listed.

PHIT	adore block with values equal to those of the set with the matching set name: 'PHIT'
Result$PHI1	adore block with values equal to those of set 'PHI1' belonging to the data set with the name: 'result'
12	adore block with the constant value 12
PHI1-PHIT	adore block with values equal to (PHI1 - PHIT), being the difference of the adore blocks with set names 'PHI1' and 'PHIT' respectively
QRCH>0	Boolean adore block containing **integer** values: equal to **1** where QRCH > 0 and equal to **0** where QRCH <= 0
(PHI1-PHIT) * (QRCH>0 && QKW1>0)	Real adore block containing values equal to **0** where QRCH <= 0 or QKW1 <= 0 and to (**PHI1-PHIT**) where both QRCH > 0 and QKW1 > 0
(RL1>TH1)?RL1:(TH1 + 0.01)	Real adore block containing values equal to **RL1** where RL1 > TH1 and to (**TH1+0.01**) where RL1 <= TH1
IF(RL1>TH1,RL1,TH1+0.01)	Real adore block containing values equal to **RL1** where RL1 > TH1 and to (**TH1+0.01**) where RL1 <= TH1

sqrt(log(cos(TX1*TH1)+1)	adore block that contains values equal to the results after evaluating the expression: $\sqrt{\log\left(\cos\left(TX1 * TH1\right) + 1\right)}$
QRI1/AREA	Specific river flux in m/d (river flux divided by node influence area)
MIN(PHIT,RP13)	Minimum value of **PHIT** and **RP13**: cut off **PHIT** at surface level
PHIT > RP13 ? RP13 : PHIT	Same as above
IF(PHIT>RP13, RP13, PHIT)	Same as above

Note:
Using Boolean expressions the result set will contain **integer** values if the expression starts with the Boolean expression and will contain **real** values if the Boolean expression is preceded with a (real) value or another expression.

Thus: (PHI1-PHIT) * (QRCH>0 && QKW1>0) results in a **real** Adore set and
(QRCH>0 && QKW1>0) * (PHI1-PHIT) results in an **integer** Adore set.

Complete expression syntax

The following table summarizes the complete expression syntax.

expression =	logical_expression
logical_expression =	relational_expression
	relational_expression '**&&**' relational_expression
	relational_expression '**||**' relational_expression
relational_expression =	additive_expression
	additive_expression '**<**' additive_expression
	additive_expression '**>**' additive_expression
	additive_expression '**<=**' additive_expression
	additive_expression '**>=**' additive_expression
	additive_expression '**==**' additive_expression
	additive_expression '**=!**' additive_expression
additive_expression =	multiplicative_expression
	multiplicative_expression '**+**' multiplicative_expression
	multiplicative_expression '**-**' multiplicative_expression
multiplicative_expression =	term
	term '*' term
	term '/' term
term =	typed_factor
	typed_factor '**^**' typed_factor
	[typed_factor '**^**' typed_factor]...
typed_factor =	factor
	'-' factor
	+' factor
	'!' factor
factor =	number
	'('expression')'
	identifier
	function(expression)
identifier =	alphanumeric string
	quoted alphanumeric string
function =	abs(..), min(..,..), max(..,..) and sign(..)
	log(..), ln(..) and exp(..)
	sqr(..) and sqrt(..)
	sin(..), cos(..), tan(..), atan(..,..), deg(..) and rad(..)
	IF(..,..,..)