triwaco

groundwater modelling software

7 Setting up and Execution of Transient Simulations

ROYAL HASKONING

Chapter 7: Setting up and execution of transient simulations

7.1 Introduction

Apart from steady-state simulations, **Triwaco** also allows the user to carry out time dependent or transient model simulations. Transient simulations are based on data defined in one of the steady-state simulation data sets, a 'Calibration' data set, a 'Scenario' data set or a 'Final' data set, and may even be based on another 'Transient' data set. In addition to the steady-state parameters some additional parameters have to be defined:

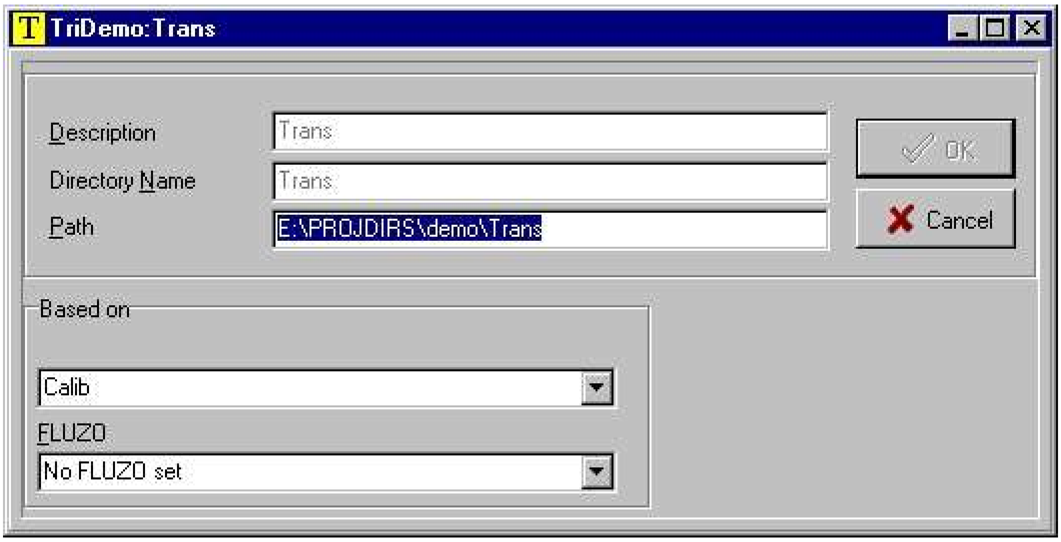
- the effective porosity of the top-aquifer (**PE**), when phreatic;
- the storage coefficient for each aquifer (**SC*i***, with ***i*** = 1,,N);
- the initial conditions for the groundwater head (**HT** and **HH*i***, with ***i*** = 1,,N).

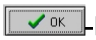
All steady-state parameters, inherited from another data set, may be replaced by time dependent or transient parameter sets. For example, where the water level **HR1** of the rivers is essentially constant in time for steady-state calculations, it may in fact vary in time. In that case, the inherited parameter set is replaced by a time dependent parameter set.

At the moment Transient simulations using ModFlow is not supported. To carry out Transient simulations using ModFlow contact the **Triwaco** helpdesk. Depending on the need by users this option can be incorperated in a custom made version.

7.2 Opening a transient data set

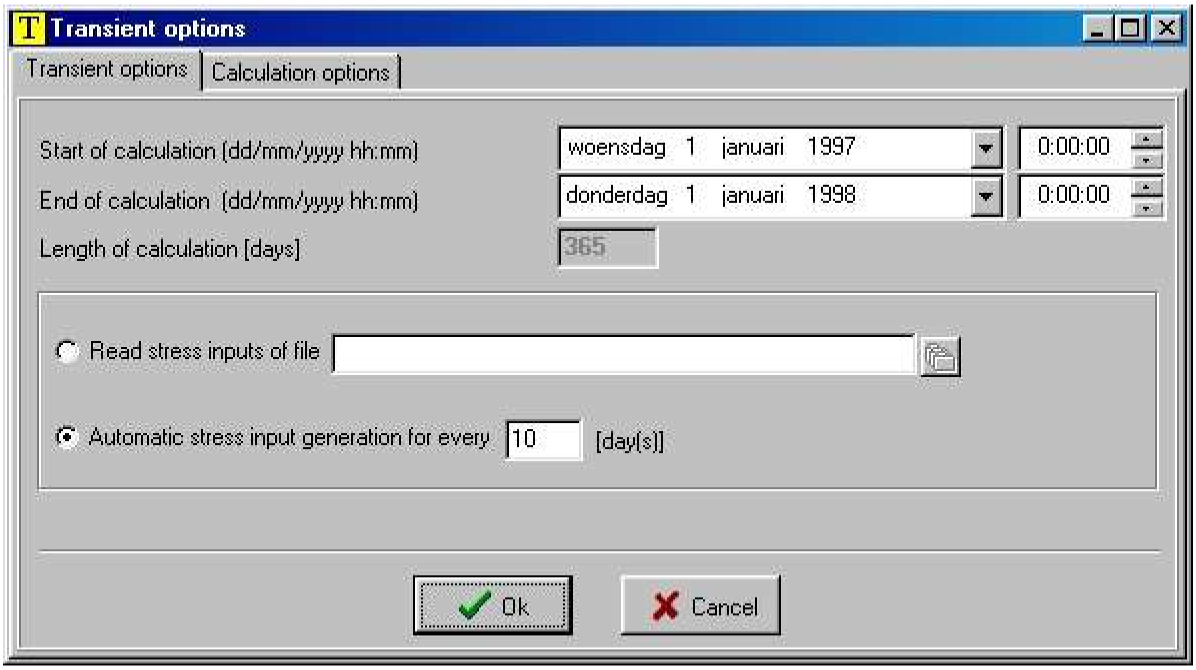
The ***'Transient data set'*** is created similarly to the other data sets by selecting 'Add' from the 'Dataset' pull-down menu and 'Transient data' from the ***'create new dataset'*** dialog window. As usual, the user is prompted to provide a description, the sub-directory name and the (unsaturated zone) data set and grid this ***'Transient data set'*** is based on. A transient data set can be based on: a 'Calibration', a 'Scenario' data set.



As soon as the user has confirmed the choices for the Transient data set, by pressing the -button, the ***Transient options window*** will appear. The window contains two tab-sheets: 'Transient options' and 'Calculation options'. The **'Transient options'** tab-sheet is used for the definition of the calculation period and the so-called '***stress input times***'. The **'Calculation options'** tab-sheet various '***simulation options***' are set.

7.3 Transient options

The tab-sheet, labeled ***'Transient options'***, is used to define the simulation period (start and end date) as well as the time step size used in the simulation.



Definition of the simulation period
The simulation period is defined by the following paramters:

The start and end times is defined in **'dd/mm/yyyy hh:mm'** and can be selected from the drop-box.

The resulting length of the calculation period is calculated and displayed in gray.

Selecting the ▼-icon will display a calendar, which enables the user to easily select the starting or end date required, scrolling back and forth with ◄ and ►.

Double clicking on the month or the year results in a list of months or a ruler for selecting the year. The hours and minutes of start and end times can be selected by using the hour rulers at the right.

Definition of stress input times
The stress input defines the times at which new parameter data input may be defined and output data are generated. The stress input times may be defined in two ways:

- *Stress input times are generated automatically*
  From the time step specified the program computes successive input times beginning at the calculation's start time. The time step has to be given in days. If hourly values are required these should be entered as a fraction of a day (e.g. 1/24 day = 0.042 day). If input is required for every decade (approximately 10-day period), as is often the case for precipitation and evaporation data, the stress input should read 10.146 days, resulting in 36 stress periods per year.

- *Stress input times are read from a so-called TIM-file*
  Choosing this option the user should specify name and location of the TIM-file to be used. This file contains a series of dates and times, in the format specified before, that should be within the range defined by the start and end time of the calculation. For each successive date and time the total time (in days) since the start time is being computed. In example TIM stress input times file is given below.

```
1
01/01/1996 00:00
31/01/1996 00:00
01/03/1996 00:00
31/03/1996 00:00
30/04/1996 00:00
30/05/1996 00:00
31/05/1996 00:00
01/06/1996 00:00
02/06/1996 00:00
03/06/1996 00:00
04/06/1996 00:00
01/07/1996 00:00
01/08/1996 00:00
01/09/1996 00:00
01/10/1996 00:00
01/11/1996 00:00
01/12/1996 00:00
```

After having defined the start and end time and the stress input times the user confirms his choice with the OK-button. The ***'Transient data set'*** is now added to the ***'project window'***. The ***'Transient data options window'*** may be reopened selecting 'Options' from the 'Transient' pull-down menu while the Transient data set window is active.

7.4 Definition of transient parameters

7.4.1 Introduction

In the transient data set two types of allocation may be distinguished:

- allocation in **space**, which is the same as for steady-state calculations;
- allocation in **time**, needed for all time dependent input parameters.

For allocation in **space** the same allocators are used for parameter allocation within the other data sets (such as ***Arpadi***, ***InvDist***, etc). Transient data set parameters for which allocation in space is sufficient are the initial heads, the effective porosity and the storage coefficient for each aquifer.

In simulations where a phreatic aquifer is used **Triwaco** automatically decides when the effective porosity (**PE**) or storage coefficient (**SC1**) for the uppermost aquifer applies. Where PHI1<RL1 (waterlevel < groundlevel) **PE** is applies, when PHI1>RL1 **SC1** applies. For all other layers a storage coefficient applies.

Because transient calculations need time dependent input, allocation in **time** is introduced here. (Allocation in **time** will also be used in the 'Unsaturated data set, chapter 13'). Allocation in **time** and in **space** is combined to generate transient parameter sets. For instance for the groundwater recharge time allocation is used to produce 10-day values and allocation in space is carried out to interpolate between points and to generate the distributed parameter sets.

In general the parameters displayed in the **'Modified parameters'** tab-sheet are essentially time dependent parameters, whereas the parameters from the **'Inherited parameters'** tab-sheet are <u>not</u> time dependent. Therefore, time allocation is carried out only for the 'modified' parameters.

For the allocation and definition of the time dependent input parameters **Triwaco** uses two different methods:

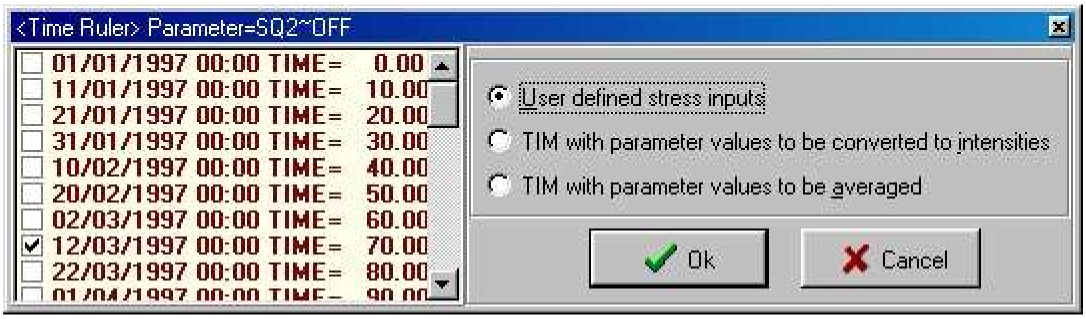
- parameter input may be given at User defined stress inputs;
- parameter input may be defined by Time series allocation.

Selecting **'User defined stress inputs'** the user can specify at which time steps changes in input parameters will take effect (c.f. change in controlled water levels of polders). For the various ways of **'time series allocation'** a so-called TIM-file is needed. Time series allocation is introduced to be able to enter parameter data, which are available in time series (c.f. meteorological data).

7.4.2 Definition by user defined stress inputs

At the left-hand side of the **'Time ruler window**' the possible stress input times are presented as 'time boxes'. Next to the time boxes the date, time and number of time step is given.

The tick-boxes in the time ruler can be checked . Whenever a box is checked the input parameter will be loaded, during the transient calculation, at the corresponding time. The first time box is checked by default, because most of the parameters are needed from the start of the calculation. If a parameter has to be used for the whole calculation, it is not necessary to check all the time boxes; in fact this parameter is treated as a steady-state input parameter. In this case it is sufficient to only check the first time box.



The groundwater flow simulation program reads parameters successively; once a parameter value is loaded this value can only be changed overwriting it with a new set of parameter values, which will be done only at input and output times mentioned in the input file. Hence, the parameter can not be switched off using the **'Time ruler'*!***

To be able to change parameter values using the ***'Time ruler'*** a special parameter name is introduced, consisting of the standard **Triwaco** parameter name followed by a '**~**' and a descriptive text string. The groundwater flow program interprets the name as if only the part preceding the '**~**' existed.

For instance, changing abstracion rate or turning off an abstraction well can be accomplished by using the following parameter names: 'SQ2**~ON**' and 'SQ2**~OFF**'.

The '**~**' in the parameter name causes the simulation program to neglect the remaining part, thus using the parameter SQ2**~ON**' for the parameter **SQ2** when it appears in the input file. Similarly, the program uses 'SQ2**~OFF**' if this parameter name is found in the ***flairs.fli*** input file. The values, however, are read from the proper parameter files: RP5**~on**.ado and RP5**~off**.ado.

Thus, using the ***'Time ruler'*** one can specify at which (input) times the parameter value should switch from 'SQ2**~ON**' to 'SQ2**~OFF**', vice versa. If the parameter 'SQ2**~ON**' is loaded at e.g. 01/01/1997, by checking the appropriate time box, the parameter's values will be overwritten if the time box for 'SQ2**~OFF**' is checked at 12/03/1997 . Thus, wells in aquifer 2 are active (when defined in SQ~**ON**) from 01/01/1997 until 12/03/1997, and well defined to be inactive (when defined in SQ2~**OFF**) from 12/03/1997 onwards.

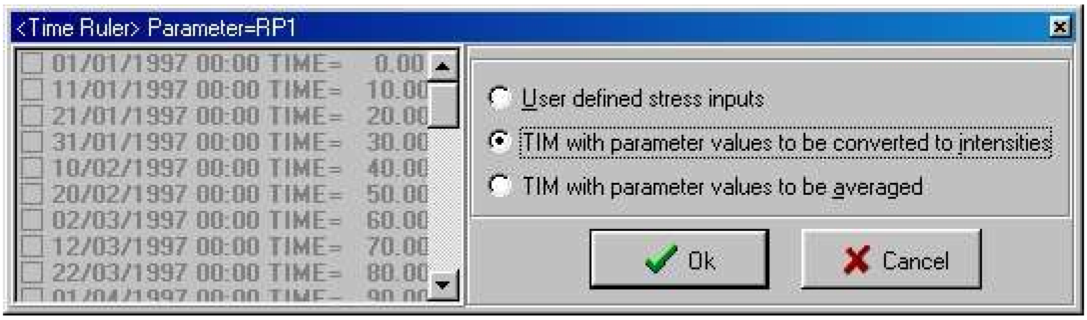
Once the stress input times are defined in the ***'Time ruler'***, the parameter can be allocated in the same way as in the ***'Calibration'*** or the ***'Scenario'*** data sets. Using the option ***'user defined stress inputs'*** thus results in one single adore set for the parameter considered.

Transient parameters that do not change with time, like the effective porosity (**PE**), the storage coefficient

(**SC1**, **SC2** etc.) and the initial conditions (**HT**, **HH1** etc.), can best be introduced using this option with the first time box of the ***'Time ruler'*** checked. For the definition of the initial conditions most often the calculation results of a steady-state data set will be used. This is simply done using the expression allocator and referring to the Result parameter in question. For example, if the initial conditions for the top-system have to be read from the ***'Calibration dataset'*** **"calib1"** one should define the parameter **HT** by the expression: **calib1$PHIT**. See also chapter 5.1.5 for the definition of initial head parameters.

7.4.3 Definition by time series allocation

The ***'Time series allocation'*** option is introduced to define input data given in a time series (e.g. meteorological data, abstraction rates or surface water levels). Time series allocation is only possible for those parameters displayed in the ***'Modified parameters'*** tab-sheet.



The ***'Time series allocation'*** type has to be activated at the right-hand side of the ***'Time ruler'*** of a parameter. Three types of ***'Time series allocation'*** (TSA) are available:

TIM with parameter values to be converted to intensities	The input times in the TIM-file do **not** correspond with the stress input times. Parameter values are given as total for the successive time periods and should be converted to an average intensity for the stress input time period. *Example*: precipitation values for a month, input required in daily values.
TIM with parameter values to be averaged	The input times in the TIM-file do **not** correspond with the stress input times. Parameter values are given for successive times or time periods and should be converted to an average value for each stress input time period. *Example*: Surface water levels at given times, input required a constant level each stress input time

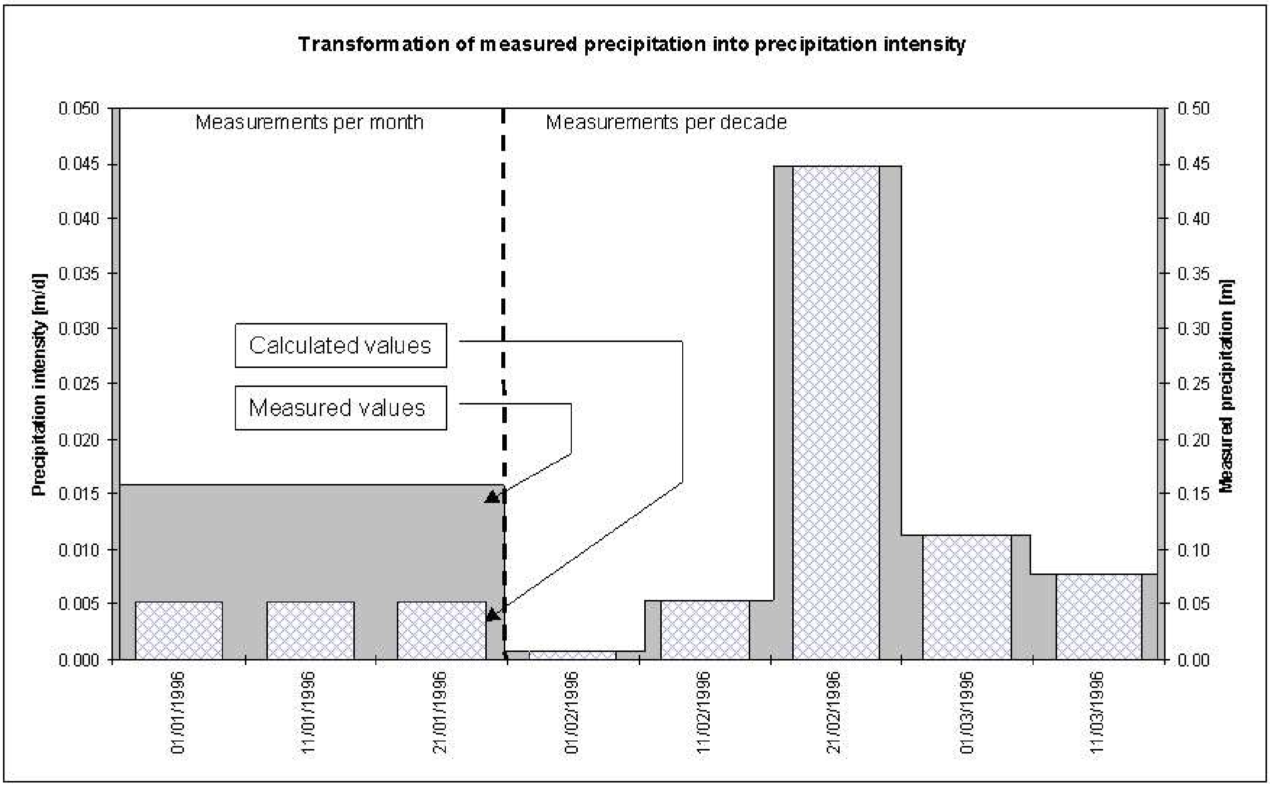
If one of the ***'Time series allocation'*** types is switched on ***'User defined stress input'*** is automatically switched off and the stress-input list at the left-hand side of the ***'Time ruler'*** window is disabled (graying it out). There is no need to check the time boxes, because input is generated for every stress-input. The stress-input times are computed by the program from the start and end time and from the 'stress input in days' defined in the ***'Transient dataset options window'*** or read from the TIM-file specified

Transient parameters to be allocated with the ***'Time series allocation'*** option should be defined by a ***map*** file and a time-dependent parameter file, the so-called TIM-file. A TIM-file (with extension *.***tim***) has to be generated by the user, which is easily done using a spreadsheet program and a text editor.

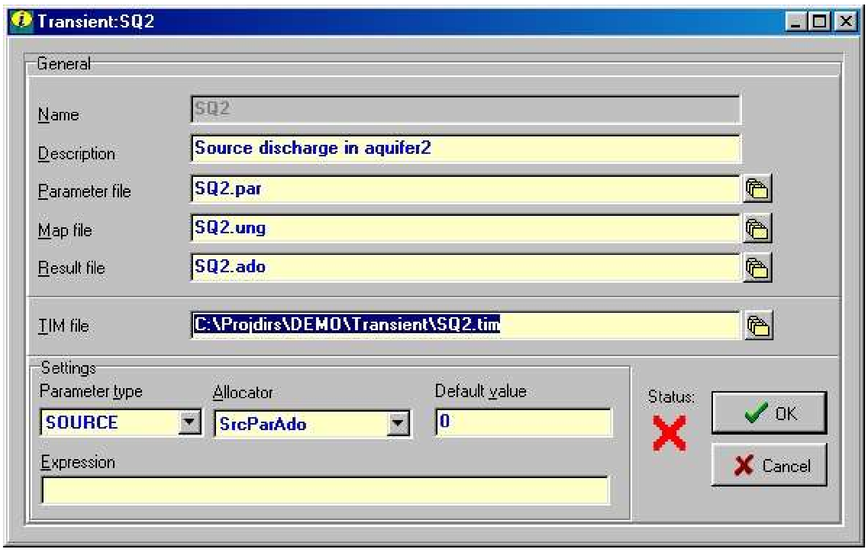
An example of a TIM-file is displayed on the right. Every set consists of the ID of the object in the map file and a number of lines containing a date, time and a parameter value. Every set is closed with the label 'END' and the total TIM-file is closed with an additional label 'END'.

```
1
01/01/1996 00:00   0.00605
11/01/1996 00:00   0.00605
21/01/1996 00:00   0.00370
11/11/1997 00:00   0.02255
21/11/1997 00:00   0.01245
01/12/1997 00:00   0.03895
11/12/1997 00:00   0.01380
21/12/1997 00:00   0.02320
01/01/1998 00:00   0.03665
END
2
01/01/1996 00:00   0.00605
01/01/1998 00:00   0.03665
END
END
```

The time interval defined in the TIM-file does not have to be equal to the stress input time step. The time series allocator implemented in **Triwaco** interpolates between successive time intervals and adds input values, if needed, to generate the proper values for each stress-input time step. The start and end of the calculation, defined in the ***'Transient dataset options window'***, should be within the range defined by the first and last time of the TIM-file. Parameter values in the TIM-file should be given in the appropriate units (e.g. monthly values of precipitation should be in meters). The output data sets, after time series allocation, are given as intensity (e.g. precipitation intensity is computed in meters/day) or given in the same units as the TIM-file, depending on the allocation type selected.



The TIM-file has to be defined in the ***'Transient parameter information'*** window, which can be accessed selecting 'Info' from the 'Parameter' pull-down menu. The TIM-file can be edited selecting 'View' 'TIM file' from the 'Parameter' pull-down menu or 'View/Edit Tim File' from the parameter pop-up menu.
A time series parameter input file (or TIM-file) can easily be generated using a spreadsheet program (using the MS Excel program one should use format: custom; dd/mm/yy hh:mm to properly format the date and time columns). After the last 'END' the program expects a '**CARIAGE RETURN**' or an '**ENTER**'.



As can be seen from the example TIM-file, a time series and corresponding parameter values should be defined for each object in the map file (each having a unique ID). After ***'Time series allocation'*** is selected and the TIM-file is defined, allocation can be carried out.

All usual methods for allocation in space can be used in combination with the time series allocation. In fact the TIM-file replaces the usual par file. Prior to the actual allocation, one can select 'Test Generate TSA file'. An intermediate file is created in which the result of the conversion to average values or intensities for each stress input time is given. Selecting 'Allocate' from the 'Parameter' pull-down or pop-up menu runs the actual allocation program and the result is written to a file containing ***multiple adore sets***. The parameter names of the various adore sets consists of the 'usual' parameter name followed by the text string ",TIME=" and the stress input time in days, relative to the start time of the calculation. Thus, the first adore set is labeled "***par***,TIME=   0.00", with for ***par*** the standard **Triwaco** parameter name.

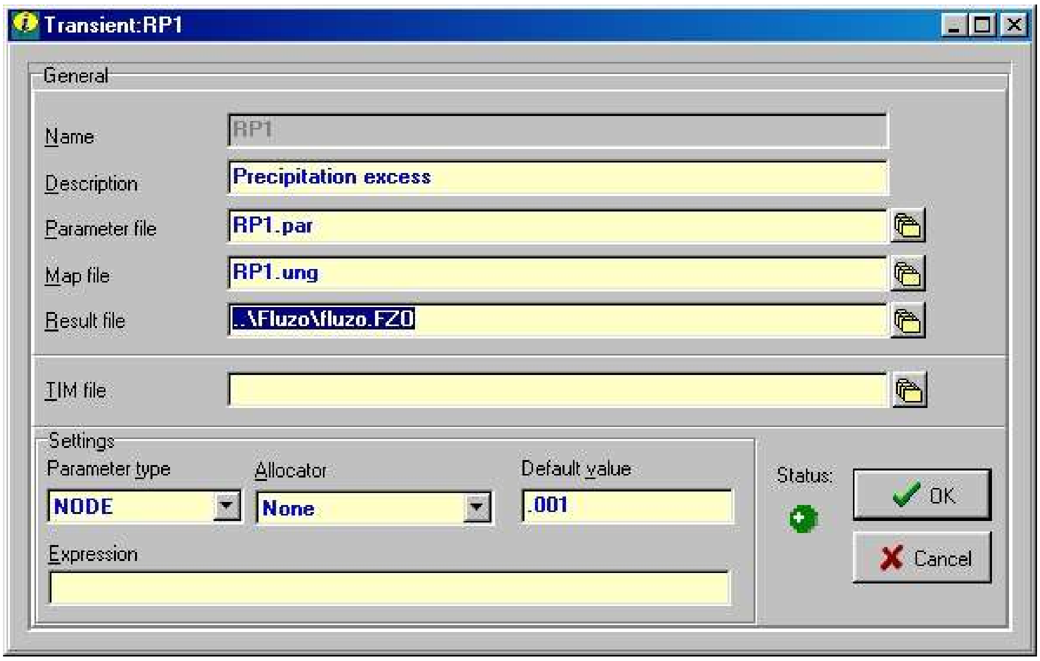
Note *that if any one of the start or end time of calculation or the stress input period is changed (in the* ***'Transient dataset options window'****) all time dependent parameters in the* ***'Modified parameters'*** *tab-sheet have to be allocated again.*

7.4.4 Definition of precipitation excess from Fluzo

For Transient simulations often time-dependent groundwater recharge is required. The module **FLUZO** (explained in more detail in chapter 12, not yet available) simulates groundwater flow in the unsatureted zone and determines the effective groundwater recharge. The effective groundwater recharge is one of the parameters of the topsystem. The effective groundwater recharge is written to a file named **fluzo.fzo**. The parameter RP1 is used to define the effective groundwater recharge in the Transient data set.

The transient groundwater recharge has to be defined in the ***'Transient parameter information'*** window, which can be accessed selecting 'Info' from the 'Parameter' pull-down menu. The fluzo result file fluzo.fzo has a the standard adore file format. Select fluzo.fzo from the Fluzo directory (search for *.fzo). The definition of RP1.ung, RP1.par can be neglected. Next activate the ***'Time series allocation'*** type 'TIM with parameter values to be converted to intensities' at the right-hand side of the ***'Time ruler'.*** There is no need to allocate this parameter since the resulting groundwater recharge is already in the appropiate ***adore*** format (fluzo.fzo-file). To prevent from accidental allocation select None as allocator.

It is important to realise that the simulation period (start and end-date) as well as the timestep size have to be identical for both the Fluzo simulation and for the Transient simulation.



7.5 Definition of a transient calibration file

If a **calibration file** (**calib.chi**) is present, **Triwaco** automatically compares calculated hydraulic heads with the data from observation wells. After comparison **Triwaco** will calculate the average deviation, the average absolute deviation, the squared average deviation, the minimum deviation and the maximum deviation. To view or edit the calibration file select 'Calibration'|'Calibration'|'View/Create Input' from the pull down menu. The input file has a fixed format described in chapter 10. The output of the calibration can be viewed as table ('Calibration'|'Calibration'|'View Output') or as a background map in **Triplot** ('Calibration'|'Calibration'|View Map'). A comprehensive description on the usage of the calibration (calib.chi) file is given in chapter 10.

7.6 Definition of a time series output file

Time series of simulated groundwater heads for user defined locations, defined by its coordinates, may be optained by defining a graphnode input file. The resulting graphnode output file is a comma delimited file and can be imported in a standard spreadsheet program to generate time graphs of groundwater heads. For all locations after each successive iteration step calculated results are written to the output file.

The graphnode input file is defined in the ***Transient options window*** under the **Calculation options** tab-sheet. The graphnode input file is defined as a standard **Triwaco** map file, i.e. ungenerated graphnode.ung file, by which user defined points are defined (see below) or a calib.chi file as used in a steady state simulation.

```
1     50744.945     408565.25
2     51637.154     407708.17
END
```

The output file contains a short descriptive heading of four records, the title for each output parameter and a series of records with the values for the output parameters. The example shown below gives the output for one user defined location for a model with one aquifers (A01). The values for the (phreatic) head in the top-system is found in the column for A00. Example Time series output file: Graphnode.out

```
Output of Flairs   version 3.1 may2003

"Time series output program FLAIRS"
"each column has heading of form <nnnnnnAaa>"
"<nnnnnn> = node number; <aa> the aquifer number"
"e.g. 012345A13 -> head aquifer 13 in node 012345"
      "TIME","000000A00","000000A01","000000A00","000000A01",
 0.0000E+00, 0.0000E+00, 1.0000E+02, 0.0000E+00, 1.0000E+02,
 5.0000E-01, 5.2291E+02, 8.8482E-02, 5.2291E+02, 3.5314E-01,
 1.5000E+00, 5.2291E+02, 8.8482E-02, 5.2291E+02, 3.5314E-01,
 3.5000E+00, 5.2291E+02, 8.8482E-02, 5.2291E+02, 3.5314E-01,
...
```

To generate time series for an arbitrary result parameter, transient simulation result or time dependent input parameter an auxiliary program is available (**mikado**). The program reads a standard **Triwaco** calibration file (**calib.chi**, for a steady-state thus without the time of observation) and the ***flairs.flo*** output file and creates an output file with, for each output time, the parameter name (e.g. "PHI1,TIME: 10.0000") followed by the calculated parameter values at the location of the observation wells.

```
"Block" "00PB000002" "00PB000003" "00PB000004"
"PHI2,TIME:    10.0000" 0.724286 0.668369 0.606561
"PHI2,TIME:    20.0000" 0.713306 0.660017 0.600011
"PHI2,TIME:    30.0000" 0.702302 0.651539 0.593338
"PHI2,TIME:    40.0000" 0.725979 0.672701 0.610671
```

7.7 Simulation options

The sheet, labeled ***'Calculation parameters'***, displays the parameters for the definition of the calculation scheme:

Iteration options
In this section the number of inner and outer iterations are set, as well as the criterion for convergence and the value for the relaxation factor. These parameters may be adopted from steady-state calculations in a ***'Calibration'*** or ***'Scenario dataset'***.

Description	Function
Inner iteration	Sets the maximum number of inner iterations
Outer iteration	Sets the maximum number of outer iterations
Convergence	Sets the criterion for convergence ε
Relaxation	Sets the relaxation factor $(\zeta \leq 1)$

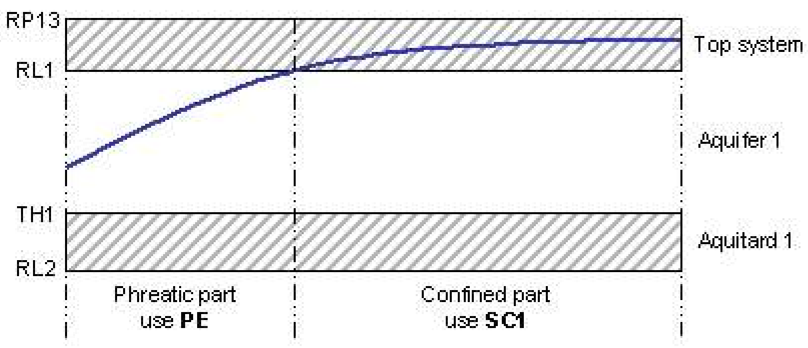
Print options
In this section the user may specify the parameters that will be written to the output file. For each layer the desired output has to be checked ☑. By default all output parameters will be printed. Output is placed in the ***flairs.flo*** file using the ***Adore*** format.

Points for time lines
In this section the graphnode input file is defined. The use and application is explained in the previous section paragraph 7.6.

Maximum allowed change in groundwater head per time step
This parameter defines the way simulation program determines the iteration time step size. The default value for this parameter is set to 0.25 m. However, in case of strongly varying surface water levels (during a flood) one might choose to increase this value to e.g. 5 m.

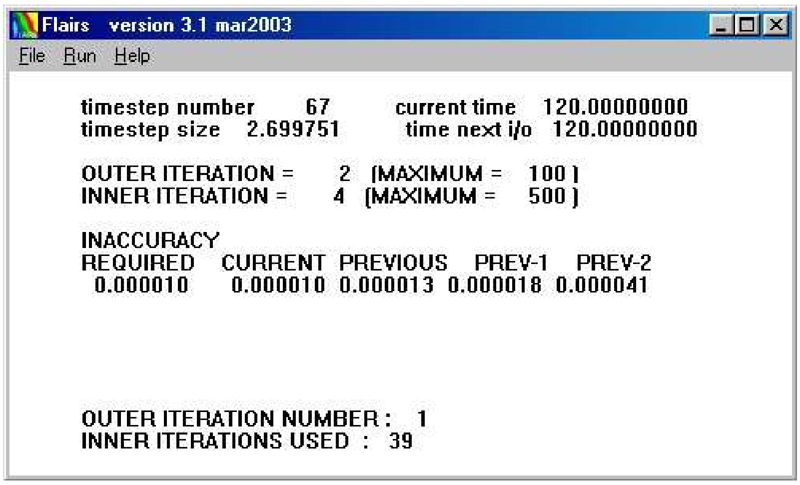
Initial time step size
The initial time step size is used by the simulation program every time new (time dependent) data is read. The default value for this parameter is set to 0.1 day. However, in case of a strongly varying situation (temporary abstraction from wells) one might choose to decrease this value to ensure a more stable calculation process. This however is not a necessity since the simulation program will automatically choose the best value during the simulation process.

Effective porosity vs storage coefficient
In simulations where a phreatic aquifer is used **Triwaco** automatically decides when the effective porosity (**PE**) or storage coefficient (**SC1**) for the uppermost aquifer applies. Where PHI1<RL1 (waterlevel < groundlevel) **PE** is applies, when PHI1>RL1 **SC1** applies. For all other layers a storage coefficient applies.



7.8 Executing the transient simulation

Selecting 'Generate input' from the 'Transient' pull-down menu will create the main input file for transient groundwater flow calculations. This file has by default the name ***flairs.fli***. The file differs from the steady-state input files by different calculation flags in the second line of the input file. Moreover, at the end of the input file a number of records are added to define the stress-input and the corresponding time dependent parameters for the transient calculations. A complete overview of the flairs.fli input file is given in chapter 5. Selecting 'Run simulation' from the 'Transient' pull-down menu will start the transient groundwater flow computations.



After termination of the computations, the results may be viewed graphically using **TriPlot**. The calculation process and the results are summarised in the output files: ***flairs.flg***, ***flairs.flp*** and ***flairs.flo*** described in chapter 5.2.6

All output is gathered in one and the same ***flairs.flo*** output file, and **no 'Result parameters'** tab-sheet will appear. Output is only generated at the stress-input times for which input data is defined. If a user wants to create more output times, one and the same parameter must be loaded at the desired times.

7.9 Viewing output results

Selecting 'View' 'Results' from the Calibration pull down menu starts the graphical presentation program **TriPlot,** loads the grid information and displays the layout of the model area. Alternatively, the user can select one of the parameters from the 'result parameters' sheet and viewing the parameter separately selecting 'View' 'Adore' from the Parameter pull down menu or 'View Adore file' from the pop-up menu (right hand mouse button). Adding other parameters (selecting 'Param' 'Load' from the **TriPlot** menu bar) gives the user

the opportunity to compare result parameters with model input parameters.
Contouring or classifying transient results is similar to that of a steady state simulation result. Creating an animation from contoured or classified results is explained in chapter 9. Creating time series graphs for any transient parameter is also explained in chapter 9.