

**triwaco**

groundwater modelling software



D Pre-processing, Post-processing and Other Tools

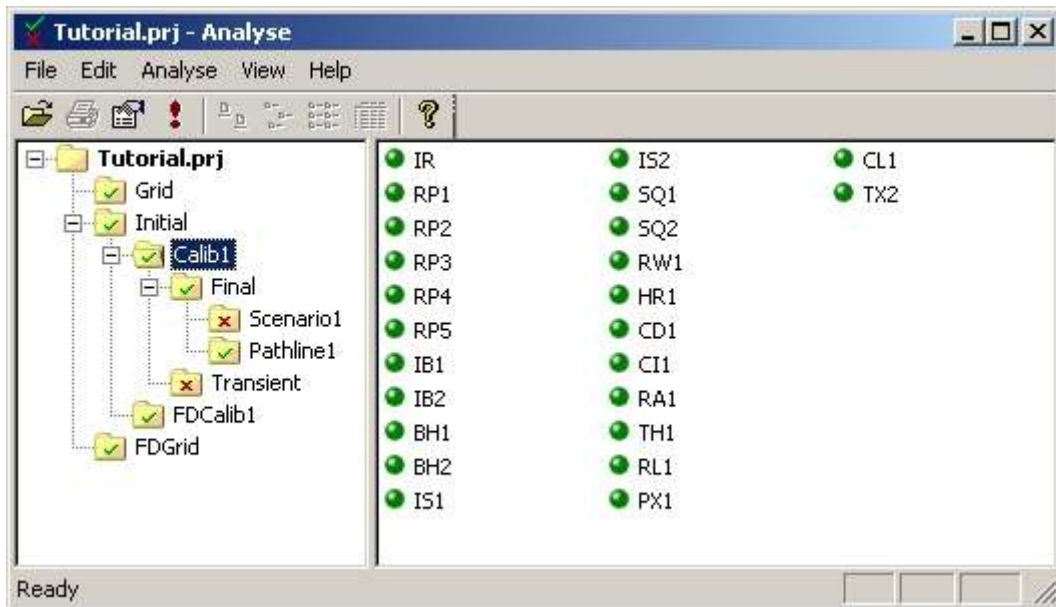
Appendix D: Pre-processing, Post-processing and Other Tools

D.1 Triwaco project analyser: Analyse.....	D-3
D.1.1 Introduction.....	D-3
D.1.2 Dataset information window.....	D-3
D.1.3 Parameter information window.....	D-4
D.1.4 Validate.....	D-5
D.2 Pre-processing Tools.....	D-6
D.2.1 Translate observation data to calib.chi: ChiZtoA.....	D-6
D.3 Post-processing Tools.....	D-7
D.3.1 Water balancing tool: BUDGET.....	D-7
D.3.2 Timeseries from transient in- and output files: MIKADO.....	D-9
D.3.3 Automatic system analysis: SYSAL.....	D-10
D.3.4 Mathematical operations on adore sets: ADOADB.....	D-11
D.3.5 Convert output files (*.ado, *.flo) to input files (*.ung and *.par): MODEL2UNG.....	D-12
D.3.6 Split large transient output files (*.ado and *.flo) in smaller size: ADOSPLITTER.....	D-13
D.3.7 Percentage of exceedence of transient in- and output files: GWSFREQ.....	D-14
D.3.8 Calculate GLG, GHG and GVG from timeseries: GLGGHGNAP.....	D-15

D.1 Triwaco project analyser: Analyse

D.1.1 Introduction

Supporting program for the analysis of the **Triwaco** project structure. The program reads information from the project- and ini-files, checks dependencies and verifies the time-stamp of the input, output and parameter files. Analyse is opened through the Tools pull down menu. A project is opened by locating the Triwaco project file (*.prj) and open.



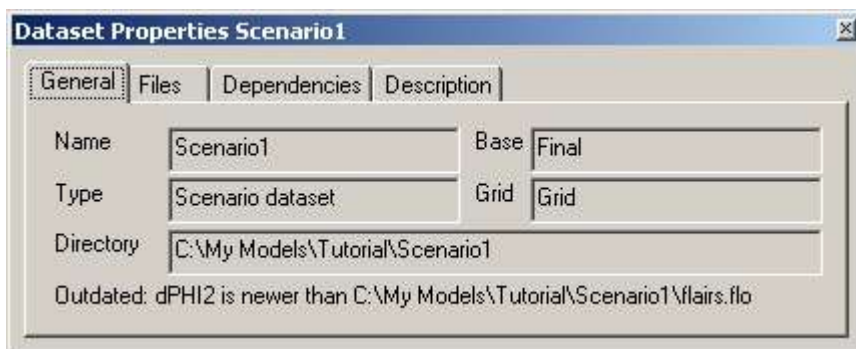
The program gives an overview of the validity of the data sets and individual parameters. Use the right hand mouse button to information windows of the datasets and individual parameters (as shown below).

D.1.2 Dataset information window

There are 4 information tabs.

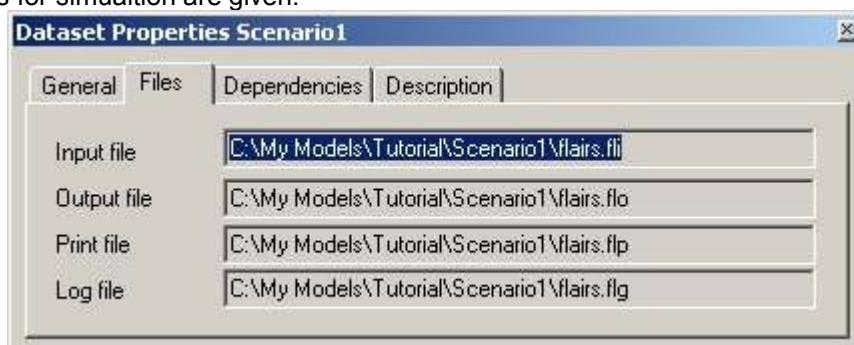
General

The general tab gives some general information about the dataset. It also gives the reason for the data set to be out of date.



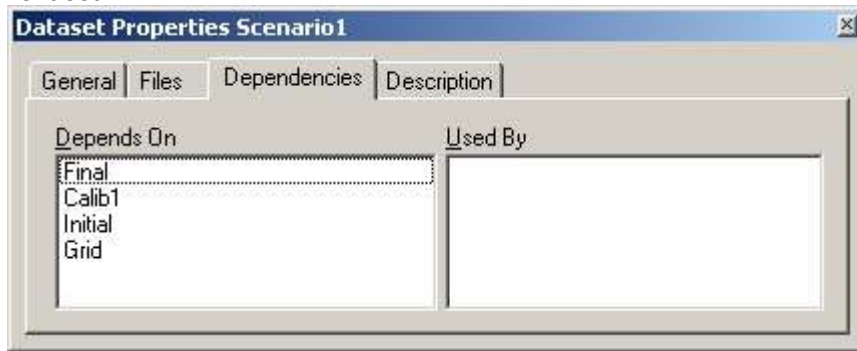
Files

In this tab the files for simulation are given.



Dependencies

This tab shows the dependencies of the dataset. It shows which data sets it depends on and which data sets depend on the current set.



Description

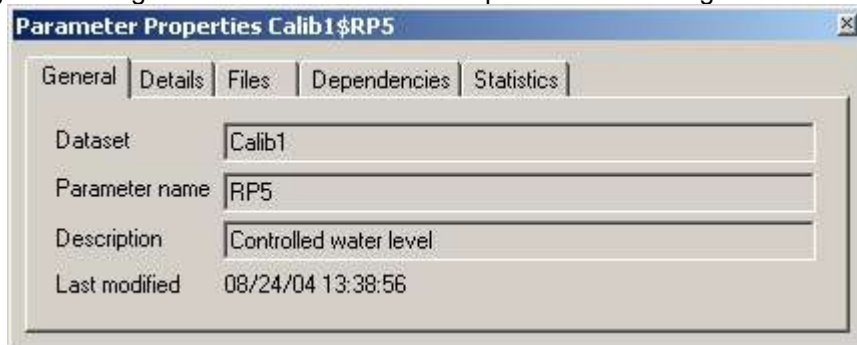
Is not used.

D.1.3 Parameter information window

There are 5 information tabs.

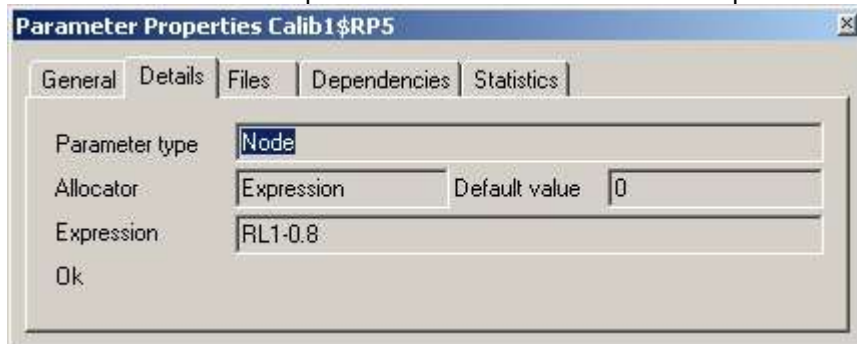
General

The general tab gives some general information about the parameter. It also gives last modified date.



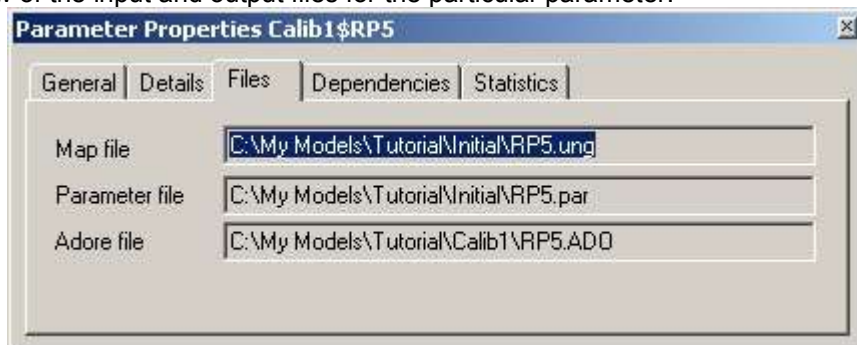
Details

The details tab gives information about the parameter. For instance the used expression.



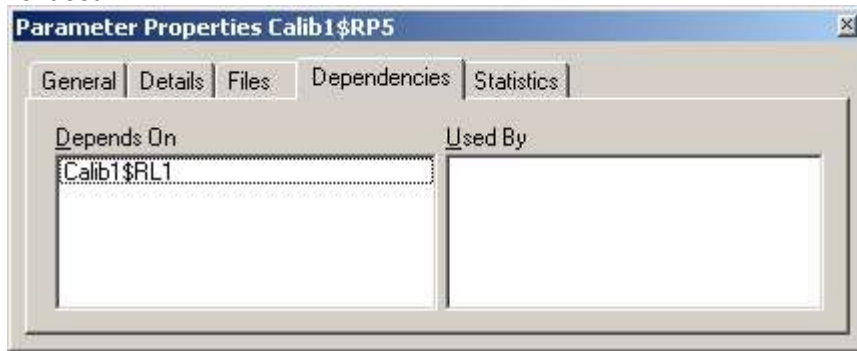
Files

Gives an overview of the input and output files for the particular parameter.



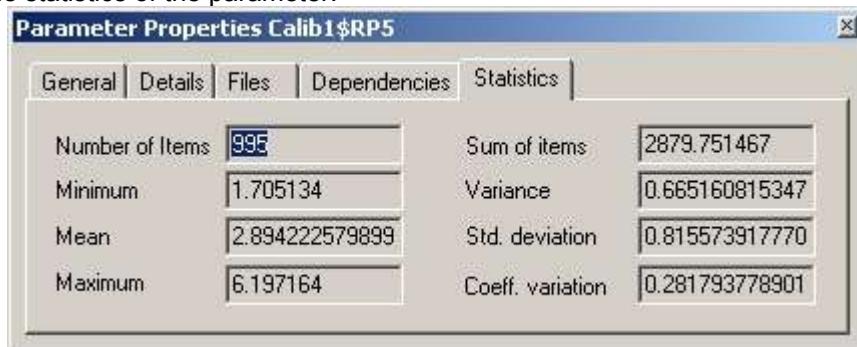
Dependencies

A very usefull tab is the dependencies tab. It shows which parameters it depends on and which parameters depend on the current set.



Statistics

This tab shows the statistics of the parameter.



D.1.4 Validate

In addition one may perform a validation of the model. Validation is carried by 'Analyse' | 'Validate'. Locate the checks file (which can be copied from the tools directory on C:\Triwaco3\Tools) with the name checks.txt. The checks.txt file contains a number of checks to be performed. The user is free to alter this file or add additional checks. Checks currently include aquifer/aquitard thickness and parameter values. After running the validation the results are written to a text-file named checks.result.txt which is automatically opened after completion. An example is given.

```
Validating dataset Calib1
Grid C:\My Models\Tutorial\Grid\Grid.teo has been loaded.

Checking thickness of layers
Thickness aquifer 1 PASSED
Thickness aquitard 1 FAILED at 995 nodes, first node = 996
Thickness aquifer 2 PASSED

Checking expressions
Verifying Top eerste pakket onder maaiveld
Evaluating RL1 < MV FAILED at 995 nodes, first node = 996
Verifying Polderpeil onder maaiveld
Evaluating PP < MV FAILED at 995 nodes, first node = 996
Verifying Rivierpeil onder polderpeil
Evaluating HR1 < PP PASSED
```

D.2 Pre-processing Tools

D.2.1 Translate observation data to calib.chi: ChiZtoA

Program for automatically assignin filterdepth of observation wells to schematized model layers.

Use

Start **ChiZtoA** form the tools pulldown menu in the **TriShell**

File | Open > flairs.fli [flairs.fli, located in data set directory with RLn en THn]

File | Grid > Grid.teo

File | Checkpoint File > Calib.chz

Input data description

Calib.chz example is present in the tools directory. Format is equal to standard calib.chi. The file calib.chz with the measured heads and filterdepth should have the following lay out:

```
[title]
[Code well 1] [X-coordinate] [Y-coordinate] [Filterdepth] [Measured value]
[Code well 2] [X-coordinate] [Y-coordinate] [Filterdepth] [Measured value]
```

Where:

Item	Description	Format**
Code well 1	Name of the observation well	[A10]
X- coordinate	X-coordinate of the observation well	[F10.*]
Y-coordinate	Y-coordinate of the observation well	[F10.*]
Filterdepth	Averaged filterdepth according to MSL (NAP)	[F10.*]
Measured value	The value to be compared with simulation result (head or head difference)	[F10.*]

Output data description

Result is a calib.chi which directly can be used for calibration.

D.3 Post-processing Tools

D.3.1 Water balancing tool: BUDGET

Budget is a program for the computation of the water balance for a standard **Triwaco** groundwater flow output file (*flairs.flo*), which may be a steady-state result or a transient result. The program reads the areas for which to compute the water balance from a map file with a number of polygons and writes the output to a print file.

Data input description

The program requires an input file (with extension *.bui) where the appropriate files are defined for input and output. An example is given below.

```
ung=budget.ung
teo=..\Grid\grid.teo
fli=..\Transient\flairs.fli
flo=..\Transient\flairs.flo
csv=budget.csv
```

Where:

Item	Description
ung*	File with polygons for which a water balance is carried out.
fli	Standard simulation input file
flo	Standard simulation output file
csv	Results are written to a standard comma delimited format file.

The polygon input file is a standard **Triwaco** map file (*.ung), containing one or more polygons with unique ID. Preferably the polygons should be completely inside the model's domain, i.e. the polygons for the water balance computation should lie within the polygon describing the model's boundary (*bnd.ung*). If a polygon is partly outside the model's domain the polygon will be clipped to the model boundary.

Output data description

The program produces a comma delimited file which directly can be opened in Excel for further processing. An example is given below.

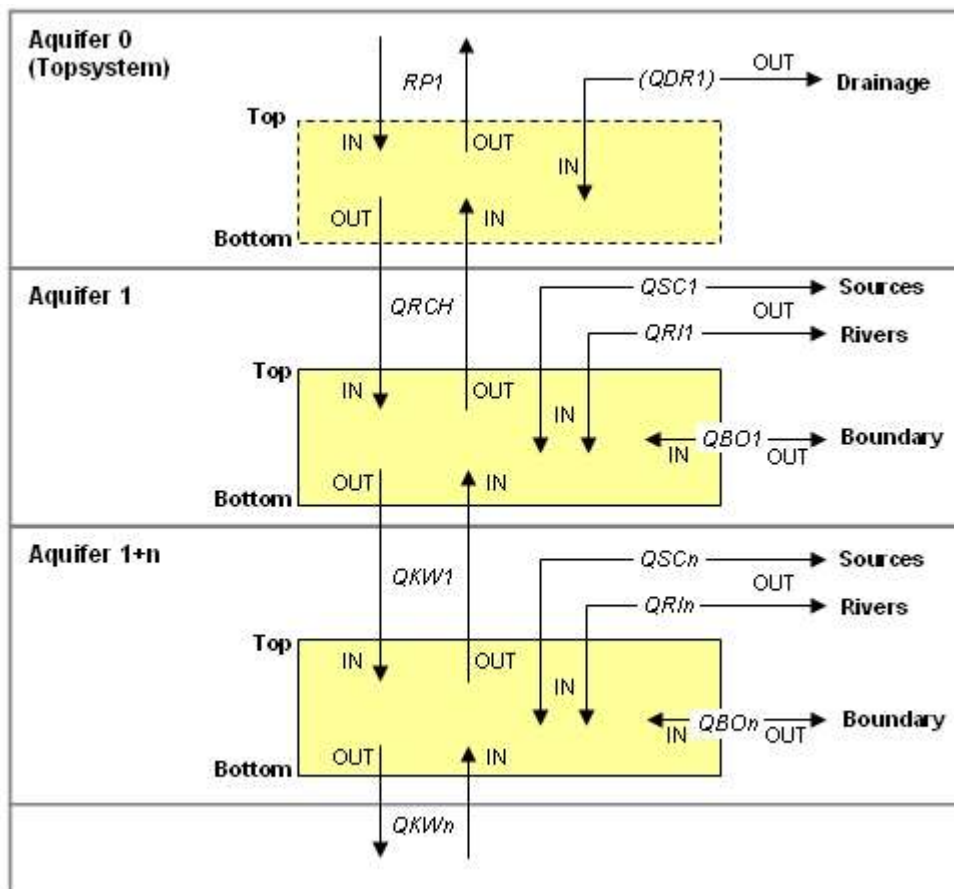
```
"NR","ID","AREA","AQUIFER","TIME","NAME","IN","OUT","SUM"
1,6,165467.76075,0,10,"Top",0,15.8023,-15.8023
1,6,165467.76075,0,10,"Drainage",262.872,0,262.872
1,6,165467.76075,0,10,"Storage",0,0,0
1,6,165467.76075,0,10,"Bottom",0,247.07,-247.07
1,6,165467.76075,1,10,"Top",247.07,0,247.07
1,6,165467.76075,1,10,"Rivers",0,0,0
1,6,165467.76075,1,10,"Sources",0,0,0
1,6,165467.76075,1,10,"Boundary",8.04869,196.259,-188.21
1,6,165467.76075,1,10,"Storage",158.509,0,158.509
1,6,165467.76075,1,10,"Bottom",0,217.368,-217.368
1,6,165467.76075,2,10,"Top",217.368,0,217.368
1,6,165467.76075,2,10,"Rivers",0,0,0
1,6,165467.76075,2,10,"Sources",0,0,0
1,6,165467.76075,2,10,"Boundary",90.9319,291.96,-201.028
1,6,165467.76075,2,10,"Storage",0,16.3394,-16.3394
1,6,165467.76075,2,10,"Bottom",0,0,0
1,6,165467.76075,-1,10,"Recharge",0,15.8023,-15.8023
1,6,165467.76075,-1,10,"Drainage",262.872,0,262.872
1,6,165467.76075,-1,10,"Rivers",0,0,0
1,6,165467.76075,-1,10,"Sources",0,0,0
1,6,165467.76075,-1,10,"Storage",158.509,16.3394,142.169
1,6,165467.76075,-1,10,"Boundary",98.9806,488.219,-389.239
1,6,165467.76075,0,20,"Top",0,12.4778,-12.4778
1,6,165467.76075,0,20,"Drainage",267.893,0,267.893
1,6,165467.76075,0,20,"Storage",0,0,0
1,6,165467.76075,0,20,"Bottom",0,255.415,-255.415
. . . . . Etc . . . . .
```

Where:

Item	Description
NR	Internal numbering of the subareas
ID	User defined numbering of subareas, which corresponds with the ID's given in the input polygon file (except when a polygon is cut by the model boundary).
AREA	Surface of clipped polygon in m2
AQUIFER*	Aquifer number*
TIME	For transient files the time
NAME**	Name of waterbalance component**
IN	Volume (m3) aquifer in
OUT	Volume (m3) aquifer out
SUM	SUM (IN + OUT)

* AQUIFER. Aquifer number is 0 for topsystem, 1, 2,...,1+n for each aquifer, -1 for the total of aquifers in that subarea.

** NAME. The different fluxes with corresponding names are shown in the picture below.



- The recharge component is equal to the standard **Triwaco** parameter RP1.
- The drainage component, in fact drainage and infiltration, is defined as the sum of QRCH + RP1 (vertical topsystem flux + recharge).
- A vertical component (recharge, top, bottom, rivers and sources) is calculated by clipping the node influence polygons with the input polygon (subarea). The vertical component for a specific node is calculated as the flux (from the simulation result **flairs.flo**) multiplied with the surface of the node influence area inside the input polygon (subarea).
- The horizontal component (boundary) is calculated as the sum of fluxes over the boundary of the input polygon (subarea). To increase the accuracy additional points are added to the input polygon on the intersection of elements (in case of Finite Elements).
- Storage is currently calculated as the balancing item of the water balance.

D.3.2 Timeseries from transient in- and output files: MIKADO

Stand-alone program for the computation of time-series of result parameters based on a calibration-type input file for user-specified locations.

Input data description

Input is a simulation output file for transient calculations (flairs.flo), a transient parameter input file (par.ado) or a series of sequential numbered input files (fl-t N .ado, with N the sequential number).

The user-specified location is read from a standard calibration file ([calib.chi](#)).

Output is written to the screen unless it is redirected to an output file.

Command line call:

```
Mikado [-o outfile] setnaam grid.teo file(s).flo
```

or, pre-WINDOWSNT

```
Mikado.exe parname Grid-file inputfilename [> outputfilename]
```

No options are available.

Example:

```
Mikado -o PHI1.OUT PHI1 ..\grid\grid.teo flairs.flo
```

or, pre-WINDOWSNT

```
mikado PHI1 ..\grid\grid.teo flairs.flo > phi1.out
```

```
mikado PHI1 ..\grid\grid.teo fl-t*.ado > phi1.out
```

Output data description

The program will create a a space delimited file which directly can be loaded into a spreadsheet (Excel).

D.3.3 Automatic system analysis: SYSAL

Sysal is a program for the determination of groundwater systems for a standard **Triwaco** influence-area output file (*infl.tro*). See chapter 11 [how to carry out an influence area calculation](#). The program reads the areas for which to determine the water groundwater system from a map file with a number of polygons and writes the output to an adore file with default name *sysal.ado*. A groundwater system is defined as a unique combination of an infiltration and seepage area.

Generally a polygon map file (sysal.ung) defines the seepage areas. Areas that are not defined by these polygons have a default value or polygon-ID equal to 0.

Sysal can be called from a batch file or directly from the command prompt:

```
Sysal.exe [polygon input file] [influence-area output file]
```

Example:

```
Sysal sysal.ung infl.tro
```

The polygon input file is a standard **Triwaco** map file (*.ung), containing one or more polygons with unique ID. The influence-area output file is a standard **Trace** output file for influence area calculations (*infl.tro*). This file contains the coordinates of start and end points of a large number of streamlines.

The program determines the polygon ID and the number start and end points of each streamline are lying in. The coordinates of the streamline's start and end points are read from the **Trace** output file: the parameter sets X-STARTPNTS, Y-STARTPNTS, X-DESTINATION and Y-DESTINATION.

The program produces an adore file (sysal.ado) containing the following sets with information on the groundwater system:

Setname	Description
STARTPOLY	Sequential number of the polygon containing the starting point
STARTID	ID of the polygon containing the starting point
DESTPOLY	Sequential number of the polygon containing the end point
DESTID	ID of the polygon containing the end point
SYSID	Code or system ID for each unique combination of STARTPOLY and DESTPOLY

The results of the system analysis may be viewed using the **Triwaco** graphical presentation tool **Triplot**, classifying the SYSID parameter.

D.3.4 Mathematical operations on adore sets: ADOADB

Post-processing tool for the manipulation of adore sets using some very simple mathematical operations.

Input from one or two ado-files; output will be written to another ado-file

Command line call:

```
Adoadb.exe operation infil parm [infil.2 parm] outfil parm [const [min max]]
```

Restrictions:

Spaces in the parameter or set name (parm) should be replaced by a '~'.

The output file (outfil) will be overwritten.

If the value one of the input files (infil or infil.2) equals -9999 the value of the output file (outfil) for operation number 1 to 12 will be set to -9999. This value is often used to indicate the absence of data ('no data' indicator).

One may choose from the following operations:

Code	Operation between ado-sets	Description
1 :	Infil(parm) * infil.2(parm)	Multiplication of two ado-sets
2 :	Infil(parm) / infil.2(parm)	Division of two ado-sets
3 :	Infil(parm) + infil.2(parm)	Summation of two ado-sets
4 :	Infil(parm) - infil.2(parm)	Subtraction of two ado-sets
5 :	Infil(parm) * const	Multiplication with constant
6 :	Infil(parm) / const	Division by constant
7 :	Infil(parm) + const	Summation with constant
8 :	Infil(parm) - const	Subtraction of constant
9 :	IF (infil(parm) > 0, const, parm)	Positive value replaced by constant
10 :	IF (infil(parm) < 0, const, parm)	Negative value replaced by constant
11 :	IF ((infil(parm) ≥ min) AND (infil(parm) ≤ max), const, parm)	Values within the range (min, max) are replaced by a constant
12 :	IF ((infil(parm) < min) AND (infil(parm) > max), const, parm)	Values outside the range (min, max) are replaced by a constant
13 :	IF (infil(parm) = -9999, infil.2(parm), infil(parm))	Replaces values from (infil) by values from (infil.2) if parm=-9999
31 :	MIN (infil(parm), infil.2(parm))	Returns the minimum of two ado-sets
32 :	MAX (infil(parm), infil.2(parm))	Returns the maximum of two ado-sets

Example:

```
Adoadb 4 flairs.flo PHIT ..\Ref\flairs.flo PHIT dPHIT.ado dPHIT
Adoadb 6 flairs.flo QRCH rech.ado Recharge~in~mm/d 1000.0
Adoadb 10 thick.ado Thick thick2.ado Thick 0.001
```

D.3.5 Convert output files (*.ado, *.flo) to input files (*.ung and *.par): MODEL2UNG

Model2ung is a program that reads information from an existing model input file (adore file or Flairs output file) and generates a map and parameter file for the corresponding parameter. The program discriminates between the various types of parameters: river, source or distributed parameter. Boundary parameters will not be recognized.

Both a map and parameter file are being generated for the whole model domain or for a window area. The location of the window area should be available as standard adore-file: window.ado. Within the window's area the value should be equal to the window's ID, outside the area the value is generally set to 0. The name of the output files should be entered on the command line without extensions. The extensions *.ung and *.par are added by the program. Apart from the parameter file, the program generates the files riv.ung and src.ung.

In case the adore parameter is converted for the whole model domain (for a non-existent window.ado file and a window ID equal to 0) the user may convert an adore-set for a distributed parameter to sets for source, river or boundary points only. Furthermore, values may be generated for a set of 'random' points (user defined points, defined in a standard map file), for which the value of the closest grid point will be assumed.

Command line call:

```
model2ung grid-file windw.ado ID infil parm outfil is ir in riv-pnt code [rndm.ung]
```

with:

wndw.ado	path and file name of the window area
ID	the ID of the corresponding window
infil	path and name of the file containing parameter values
parm	parameter set name
outfil	name of output files (without extension)
is	value all original source ID's should be increased with
ir	value all original river ID's should be increased with
in	value all original node ID's should be increased with
riv-pnt	output option for river parameters
=0	output of all river nodes
=1	output of river input points only
code	output option for conversion of distributed parameter
=0	no conversion
=1	conversion to river nodes
=2	conversion to source nodes
=3	conversion to boundary nodes
=4	conversion to user defined points
rndm.ung	map file with random points for which output is generated.

Example:

```
model2ung ..\old\grid.teo ..\old\win.ado 1 ..\old\tx2.ado TX2 TX2 0 0 0 0 0
```

D.3.6 Split large transient output files (*.ado and *.flo) in smaller size: ADOSPLITTER

ADOSPLITTER is a program to split large transient output files in smaller size. This program was created to deal with the maximum WINDOWS file size (files larger than 4 Gb).

Command line call:

```
adosplitter -g2 file.ado splitst file.ado in file.001.ado, file.002.ado, etc. Each file is max 2 Gb  
adosplitter -m700 file.ado splitst file.ado in file.001.ado, file.002.ado, etc. Each file is max 700 Mb.
```

Example:

```
adosplitter -g2 flairs.flo
```

D.3.7 Percentage of exceedence of transient in- and output files: GWSFREQ

Post-processing program for the calculation of groundwater heads (or other time dependent parameters) with a given percentage of exceedence.

Input is a standard **Flairs** output file for transient calculations (**flairs.flo**) or a transient parameter input file (par.ado). The input file should contain multiple adore sets, for successive time steps.

The program generates an output file with a number of adore sets, defining the parameter values for 0, 10, 50, 90 and 100% of exceedence and the average value.

Command line call:

```
Gwsfreq.exe set-dir out-dir infil parm outfil t0 tE code
```

The value of '**t0**' defines the start time for the computation and the value of '**tE**' the end time. These times are relative times (in days) as is customary for **Triwaco** transient calculations and should lie between the first and last time step encountered in the input file (**infil**).

Two options are available for the code: '**i**' or '**u**'. Selecting '**i**' the file '**infil**' is supposed to contain input parameters at the start of every time step. Selecting '**u**' the file '**infil**' is supposed to contain output parameters at the end of every time step.

Example:

```
Gwsfreq .\Calib .\Result flairs.flo PHIT GWL.ado 0 360 u  
Gwsfreq .\Calib .\Result HR1.ado HR1 HWL.ado 0 360 i
```

D.3.8 Calculate GLG, GHG and GVG from timeseries: GLGGHGNAP

Post-processing program for the calculation of GLG, GHG, GVG, Highest and Lowest value form a standard transient **Triwaco** output file (**flairs.flo**) or observation time series.

Observation time series

First a input file (***.prn**) has to be created containing the observation time series. An example is given below. The first row indicates the number of lines with observations dates, the second value indicates the number of columns used. For each observation four columns are used (day, month, year, head). So in the example there are 2 observation points with 281 observations.

```
281 8
14 02 1981 10.01 14 02 1981 12.42
28 02 1981 9.89 28 02 1981 12.20
14 03 1981 1e+99 14 03 1981 12.81
28 03 1981 10.00 28 03 1981 12.47
28 04 1981 9.81 28 04 1981 12.10
28 05 1981 9.85 28 05 1981 12.25
14 06 1981 9.84 14 06 1981 12.23
28 06 1981 9.89 28 06 1981 12.37
28 07 1981 9.76 28 07 1981 12.03
28 08 1981 9.69 28 08 1981 12.02
28 09 1981 9.64 28 09 1981 12.00
28 10 1981 9.92 28 10 1981 12.30
28 11 1981 10.12 28 11 1981 12.49
...
```

Then start the program and choose Prn-file.



The program will then ask to select the input file, to name of the output file and finally the start and end date for the period to be evaluated. Then simply press OK and the evaluation will start.



Standard Triwaco transient output file

To evaluate using a standard **Triwaco** transient output file (PHIT=groundwater level) simply start the the program and choose Ado-file.



The program then will ask for a transient output file, the start date (Use data after (d)) and finally the decade length. The results will be written to a standard adore file (**GHGGLGNAP.ado**) which can directly be loaded into **TriPlot**.

