

**MIKE URBAN CS**  
*'DHIAPP.INI' AND '\*.ADP' FILES*  
Reference Manual



**DHI headquarters**

Agern Allé 5  
DK-2970 Hørsholm  
Denmark

+45 4516 9200 Telephone

+45 4516 9333 Support

+45 4516 9292 Telefax

[mike@dhigroup.com](mailto:mike@dhigroup.com)

[www.mikepoweredbydhi.com](http://www.mikepoweredbydhi.com)

## PLEASE NOTE

### **COPYRIGHT**

This document refers to proprietary computer software, which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programmes is prohibited without prior written consent of DHI. For details please refer to your 'DHI Software Licence Agreement'.

### **LIMITED LIABILITY**

The liability of DHI is limited as specified in Section III of your 'DHI Software Licence Agreement':

'IN NO EVENT SHALL DHI OR ITS REPRESENTATIVES (AGENTS AND SUPPLIERS) BE LIABLE FOR ANY DAMAGES WHATSOEVER INCLUDING, WITHOUT LIMITATION, SPECIAL, INDIRECT, INCIDENTAL OR CONSEQUENTIAL DAMAGES OR DAMAGES FOR LOSS OF BUSINESS PROFITS OR SAVINGS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION OR OTHER PECUNIARY LOSS ARISING OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, EVEN IF DHI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THIS LIMITATION SHALL APPLY TO CLAIMS OF PERSONAL INJURY TO THE EXTENT PERMITTED BY LAW. SOME COUNTRIES OR STATES DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL, SPECIAL, INDIRECT, INCIDENTAL DAMAGES AND, ACCORDINGLY, SOME PORTIONS OF THESE LIMITATIONS MAY NOT APPLY TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'



# CONTENTS

## MIKE URBAN CS 'DHIAPP.INI' AND '\*.ADP' FILES Reference Manual

<b>1</b>	<b>Control of MOUSE Computations - “DHIAPP.INI” File .....</b>	<b>1</b>
1.1	Extended Explanations .....	15
1.1.1	Reporting flooding volumes and inflow limitation volumes to the Extra Result file (.XRF) .....	15
1.1.2	Variable speed pumps – output of operation .....	21
1.1.3	Grain size mobility .....	23
<b>2</b>	<b>The "*.ADP" File.....</b>	<b>25</b>
2.1	Introduction to *.ADP.....	25
2.1.1	Detailed description of data sections in the *.ADP file .....	25
2.2	General Features .....	26
2.2.1	Computing volume for a selected part of the CS network .....	26
2.2.2	Initial water level – set from any node.....	29
2.2.3	Variable headloss coefficients .....	30
2.3	MOUSE LTS - Long Term Simulations - Features .....	31
2.3.1	LTS - output of the Stop Criterion flow.....	31
2.3.2	LTS - additional output from Job List generation .....	33
2.3.3	LTS - DWF insertion between events .....	34
2.4	Link to Groundwater Model MIKE SHE.....	35
2.5	Structures .....	35
2.5.1	Control of pump behaviour & output from these .....	35
2.5.2	Radial Gates .....	38
2.5.3	Explicit weirs .....	39
2.6	Control.....	40
2.6.1	PID Sampling Period.....	40
2.6.2	Control of Coupled Pumps .....	41
2.7	MOUSE Pipe Design.....	41
2.8	Biological Processes and Pollution Transport .....	43
2.8.1	Hydrogen sulphide build-up .....	43
2.8.2	Inverted siphons.....	45
2.8.3	Sediment traps .....	45
2.8.4	Fixed sediment depth in individual conduits .....	46
2.9	Sections only Relevant for Discontinued MOUSE Package .....	47
2.9.1	Links (section only relevant for MOUSE) .....	47
2.9.2	Colebrook – White (section only relevant for MOUSE).....	48
2.9.3	Control of PID Minimum Capacity Curve (section only relevant for MOUSE) .....	49
2.9.4	Additional DWF definition.....	49
2.10	Pump Emergency Storage Estimator (section only relevant for MOUSE).....	50
2.10.1	[PUMP_ESE] Section.....	52
2.10.2	Example of Pump Emergency Storage Estimator – ADP file .....	56



## 1 Control of MOUSE Computations - "DHIAPP.INI" File

MOUSE computations are complex, and apart from input data, the results depend on default settings of a number of program parameters. These parameters may activate different formulations for the specific aspects, set the threshold or default values for variables, etc. MOUSE is delivered by a set of default parameters. Users can access and modify some of these parameters (e.g. control of variable time step, print of warnings, type of output, etc.) i.e. change the default model performance in various aspects of computations. The controllable parameters are accessible in the ASCII file 'DHIAPP.INI', located per default in the MIKEURBAN/cfg directory.

The 'DHIAPP.INI' file can be edited by any ASCII editor. An edited copy can overwrite the default version, or it can be saved into the wanted project directory under the same name. If located in the project directory it will be used for the specific project only.

Some of the parameters accessible through the 'DHIAPP.INI' file are not explained in detail since they are related to specific technical issues. The parameters which are likely to be relevant to standard MOUSE applications are self-explanatory.

**IMPORTANT!** Modifications of the original 'DHIAPP.INI' file, located in the MIKEURBAN\cfg directory will affect all MIKE URBAN projects computed with the MOUSE or MIKE 1D engine. This may be relevant for general tuning of the engines. Project-specific modifications must be introduced by copying the edited 'DHIAPP.INI' file into the current project directory.

In the following, a print of the DHIAPP.INI file is presented.

```
; DHIAPP.INI file
; Used for MIKE by DHI Product : MIKE URBAN
;
[General]
; Support for enhanced unit precision when converting units to/from US
units
;ENHANCED_UNIT_PRECISION=ON
;
; The summary file contains a summary of boundary conditions. As input
to the ;model is discretized based on the timestep the accumulated value
can either ;represent the accumulated value used in the simulation
(default) or the ;accumulated value based on input data.
;
CALCULATE_BC_SUMMARY_FROM_TS=0
;
[Mbrowser]
; MOUSE Text and Html Browser Specification
; Used for presentation of error/status/summary report after simulation.
; 'Internet Explorer' is shown as example.
; HTMBROWSER=C:\Program Files\Internet Explorer\IEXPLORE.EXE
; Other text file editors or browsers for Windows may be used.
; If none are specified, the associated viewers for the file types are
used.
;HTMBROWSER=C:\Program Files\Internet Explorer\IEXPLORE.EXE
;
; During data reading and after the simulation MOUSE generates several
LOG-files
; and a simulation summary file. In consecutive simulations, MOUSE
maintains a set of
```

```

; backup LOG- and summary files, by assigning serial numbers to the
; files. The files
; indexed "001" are the newest, and growing index numbers indicating
; older files.
; The maximum number of the backup files maintained can be controlled by
; setting
; the parameter MAX_LOG_FILE. The default value is 5
MAX_LOG_FILE=5
;
; There are 2 naming conventions when creating summary files after each
; simulation
; The summaries can either contain the result file name or the project
; file name.
; SUMMARY_NAME=1 Names the summary including the result filename
; SUMMARY_NAME=2 Names the summary including the project filename
SUMMARY_NAME=2
;
[MOUSESimLaunch]
; Application thread option. Default = 0 (No thread suspend in plot
; selection dialog)
; Thread suspend may be required in some configurations. Set
; THREADOPTION = 1 to apply thread suspend.
;THREADOPTION=0
;
[MouseRunoff]
;-----
;-----
; Section for parameters controlling simulation with MOUSE Runoff
;-----
;-----
;
; Summary table at the end of the Runoff simulation.
; This can be switched ON/OFF by the parameter:
; SummaryOutput=0 No summary
; SummaryOutput=1 Write summary (default)
SummaryOutput=1
;
; -----
; Infiltration options - Horton's equation
; -----
; It is possible to select between the former standard version of the
; Horton's equation
; and the integrated version of the Horton's equation. For details
; please refer to the
; MOUSE Reference Manual (Runoff). In addition it is possible to specify
; if
; infiltration should be active only during rainfall, or if infiltration
; should be
; active also after rainfall until surface runoff ends. Any of the four
; combinations
; can be specified by the parameter below.
;
; IntegratedHorton =
; 0 : Standard Horton. Infiltration active during rainfall and after
; until runoff ends
; 1 : Integrated Horton. Infiltration active ONLY during rainfall
; 2 : Standard Horton. Infiltration active ONLY during rainfall
; 3 : Integrated Horton. Infiltration active during rainfall and after
; until runoff ends
; Option 0 and 1 : as before
; Option 2 and 3 : new options from 20041118
; Option 0 is default in MOUSE 2004 and previous versions
IntegratedHorton=0

```



```

; In the two cases: IntegratedHorton=0 and IntegratedHorton=3 the
switching
; to regeneration of infiltration capacity is controlled by a low limit
of
; the water depth on the surface.
; The limit can be set by the entry: RUNOFF_B_DRYDEPTHMM the unit is
[mm]
; The default value applied by Runoff Model B is
RUNOFF_B_DRYDEPTHMM=0.001
; RUNOFF_B_DRYDEPTHMM=0.1

; -----
; Runoff Model B - Extra Output
; -----
; An option for writing detailed information about the hydrology
processes
; in the Runoff Model B is available. The extra output will be sent to
the
; normal CRF result file.
; Notice: This feature will produce significantly larger CRF files.
; Therefore the feature is available only for setups with <= 3
catchments.
; The extra output can be presented in MIKE VIEW. The feature is
intended
; for better understanding of the detailed processes in Runoff B,
calibration,
; test and documentation.
; It is recommended to switch the extra output OFF when producing CRF
files
; for use as input to MOUSE HD. Default is: RUNOFF_B_ExtraOutput=OFF
; The "extra output" is available in MOUSE 2004 update from 10 Dec 2004
; or later.
; RUNOFF_B_ExtraOutput=ON

;
; Improvements to the numerical scheme used in Runoff B were introduced.
; The improvements can be switched ON/OFF.
; The switches for the modifications in Runoff B engine are:
; RUNOFF_B_NEW1=ON (default)
; RUNOFF_B_NEW2=ON (default)
RUNOFF_B_NEW1=ON
RUNOFF_B_NEW2=ON
;
; Low flow limit for switching to dry period time step.
; The dry period time step will only be applied when
; the runoff from the rainfall is below the limit
; for all catchments. Default value is 0.0001 [m3/sec]
RUNOFF_B_LOW_FLOW = 0.001
;
; Runoff Model A, B and C
; Recovery rate for initial loss capacity.
; The initial loss capacity is regenerated in dry periods by a linear
recovery rate. Default value is 0.00005 [m/hour] for all models A, B
and C
; The rate can be controlled by the parameter
RUNOFF_A_INITLOSS_RECOVER
; Set the value to 0.00 if you want the Runoff model to work as in
; previous versions of MOUSE.
RUNOFF_A_INITLOSS_RECOVER = 0.00005
RUNOFF_B_INITLOSS_RECOVER = 0.00005
RUNOFF_C_INITLOSS_RECOVER = 0.00005
;
; Include evaporation in runoff model B. When ON the initial loss
capacity is regenerated

```

```

; based on the evaporation and RUNOFF_B_INITLOSS_RECOVER is not used.
Default is OFF
; RUNOFF_B_EVAPORATION=OFF
;
; When carrying out calculations with the SQR module of MOUSE TRAP or
when using MOUSE Online it can be
; relevant to switch off the jump between rain events meaning that
calculations will also be carried out
; for dry weather periods.
;RUNOFF_B_Variable_dT=OFF
;RUNOFF_A_TimeJump=OFF
;
; RMSE_Method parameter = 1 will calculate RMSE in peaks dividing with
the number of peaks afterwards.
; RMSE_Method parameter = 2 will calculate RMSE in peaks without taking
the average value afterwards. (Default = 2)
; RMSE_Method parameter = 3 will calculate the sum of the absolute
difference for peak maxima, i.e. 1 value per peak.
;RMSE_Method=3
; The WBL_Method parameter = 2 will calculate WBL in peaks without
taking the average value afterwards. (Default = 2)
; The WBL_Method parameter = 3 will calculate |Qsim-Qmeasured| in peaks
for all values above the peak threshold.
;WBL_Method=3
;
; Switch for using the total area or the reduced area as basis for the
mass of sediment that can be washed off.
; Default is use the reduced area = TRUE
;SRQ_USE_REDAREA=FALSE
;
[MouseRunoff_C]
;-----
; Section for parameters controlling simulation with MOUSE RunOff C
;-----
;-----
; MOUSE Runoff Model C (C1)
; NL_CATCHMENT_GROUPING = 1
; For Runoff Model C1 the result file can be reduced significantly
; by allowing catchments to be "grouped". The principle is that for
; all catchments which have a common parameter set - the resulting
; hydrograph can be scaled form a 'unit' hydrograph. Only one
; hydrograph is saved in the CRF result file. When the CRF file
; later is used as input for the HD simulation, then all
; hydrographs will be generated automatically by multiplying the
; 'unit' hydrograph.
; The feature is only implemented for the C1 variant.
; NL_CATCHMENT_GROUPING = 0 (Default)
; No grouping of catchments. All individual hydrographs will be
; written as results to the CRF file.
NL_CATCHMENT_GROUPING = 0
;
; NL_CONCEPT_DRYDEPTH
; This parameter can be used to control the minimum depth of water
; in the reservoir at which the simulation will switch to DRY
; conditions. The unit is [mm]. The default value is 0.001 mm.
NL_CONCEPT_DRYDEPTH = 0.001
;
; The Runoff model variant C2 (French Hydrology concept) uses the
Desbordes formula
; for calculating the Lag Time when this parameter is not user
specified.
; Prior to Release 2012 of MIKE URBAN a constant in the formula was set
to 0.3175 without

```

```

; possibility of changing the parameter. From Release 2012 the default
; value of constant is set to 0.494
; in order to be in compliance with scientific documentation for the
; applied hydrologic model.
; It is possible to switch calculations back to using the former value
; of the constant by using the entry below:
; RunoffC2LagTimeConstSwitch = 0   This will apply the former value =
; 0.3175
; RunoffC2LagTimeConstSwitch = 1   This will apply the new value = 0.494
;
RunoffC2LagTimeConstSwitch = 1
;
[MouseNam]
;-----
;-----
; Section for parameters controlling simulation with MOUSE RDII
;-----
;-----
;
; RDII switching criteria: RDII+A/B --> RDII
; =====
; The criteria for switching FROM the detailed surface runoff
; simulation with RDII+A or RDII+B TO only RDII in dry periods
; can be controlled by the following parameters:
;   A_STOP_FLOW=          default value is 0.001 m3/sec
;   B_STOP_FLOW=          default value is 0.001 m3/sec
; The switch will take place when the sum of the FRC runoff
; from all catchments falls below the specified limit
; AND
; the actual rainfall is below the rain limit specified below.
A_STOP_FLOW=0.001
B_STOP_FLOW=0.001
;
; RDII switching criteria: RDII --> RDII+A/B
; =====
; The criteria for switching TO the detailed surface runoff
; simulation with RDII+A or RDII+B FROM only RDII in dry periods
; can be controlled by the following parameters:
;   A_LIMIT_RAIN = default value is 0.00001 my-m/sec
;   B_LIMIT_RAIN = default value is 0.00001 my-m/sec
; The switching will happen when the rainfall exceeds the applied
; limit.
; NOTE: In periods with snow cover the switching criteria are
; over-ruled until the snow reservoir is empty.
A_LIMIT_RAIN=0.00001
B_LIMIT_RAIN=0.00001
;
; RDII hot start
; =====
; RDII hot start was enhanced by writing additional storage depths used
; in
; overland flow computation to the .NOF result file.
; Due to this improvement, older *.NOF files cannot be used for hot
; start. It
; is recommended to re-do the simulation and produce the new improved
; files.
; For backward compatibility this can be switched off by setting
; the parameter NAM_HOTSTART_VER=0
;
; Default setting is: NAM_HOTSTART_VER=1
NAM_HOTSTART_VER=1
;
; RDII+A use of reduction factor
; =====
; RDII has from MOUSE 2002 version the possibility for using

```

```

; the 'Reduction Factor' found in Model A input data. The reduction
; factor is applied to the Fast Runoff Component (FRC).
; Before MOUSE 2002 the Reduction Factor was ignored.
; It is possible to switch the application of the reduction factor
; On/Off by the following entry:
; Use_ModelA_RedFactor = ON
; Use_ModelA_RedFactor = OFF
; The default is 'ON' in MOUSE 2002 if the entry is omitted
;
; RDI+B reduce size of result files
; =====
; To reduce the size of the CRF result file RDI+B will save results with
; SRC time step in periods with snow cover and no precipitation.
; Valid for MOUSE 2004 version.
; Before RDI+B saved results in these periods with the FRC time step.
; The following entry can be used to switch back to the previous result
; saving method.
; RDIBSNOWRESULTDETAILED=ON
;
; Switch for writing the NOF file in a RDI solo simulation. Default is
writing the NOF file.
WRITE_NOF_FILE=ON
;
[MouseHD]
;-----
; Section for parameters controlling simulation with MOUSE HD
;-----
;
; The MOUSE HD simulation produces a text file with detailed information
about water volume generated in dry parts
; of the network during the simulation period. The feature will also
generate a MIKE URBAN Selection file (*.mus),
; including the links and nodes reported.
; The following parameters control the output:
;   HDSummaryGeneratedWaterReport : Switches the reporting feature ON
(=1 default) or OFF (=0)
;   HDSummaryGeneratedWaterLowLimitLink : Only the links with water
volume generated higher than this threshold are reported (default =
0.100)
;   HDSummaryGeneratedWaterLowLimitNode : Only the nodes with water
volume generated higher than this threshold are reported (default =
0.010)
;   HDSummaryGeneratedWaterDecimals      : Number of decimals for
reported volumes (default=3)
; The unit of the threshold is M3 for summary in SI units and CFT for
summary in US-units.
;
HDSummaryGeneratedWaterReport=0
HDSummaryGeneratedWaterLowLimitLink=0.100
HDSummaryGeneratedWaterLowLimitNode=0.010
HDSummaryGeneratedWaterDecimals=3
;
;
; ----- Pressure Mains error check -----
; Strict checking of Pressure Main sub-networks was introduced with
Release 2009 Service Pack 4.
; The error check allows only PM sub-networks configured of PM links and
terminated by a PM Tail Node.
; This is the prerequisite for the implemented calculation algorithms.
(See manuals for details)
; -----
; Advantage: The strict error checks helps identify and correct
unintended network

```

```

;           configuration which otherwise violate the computation
results.
;   Disadvantage: Some configurations which may have produced acceptable
results will not be allowed.
;   Notice: DHI wants to analyze these cases in details before making it
part of the default allowed configurations.
;   -----
;   For the cases listed below you may loosen the strict error check.
;   When this is done you also accept that some computational algorithms
are being used outside
;   the original prerequisite for their use. Computational results should
be checked carefully.
;   -----
;   Allow valves in PM sub-networks:
;   By setting the entry below = 1 then valves will be allowed as part
of PM sub-network configurations.
;   You will have a warning instead of the error message.
AllowPMLinksConnectToValves=0
;   Allow PM links to discharge to nodes with pumps without setting the
node as Tail Node:
;   By setting the entry below = 1 then this connection will be
allowed
;   You will have a warning instead of the error message.
AllowPMLinksConnectToPumps=0
;   -----
;
;   When loading large network models with very large number of catchments
connected
;   it may be necessary to adjust the buffer size for the loading of
Runoff results
;   time series from the CRF file. Reduce the buffer size if you get an
OUT OF MEMORY
;   error message during loading of data.
;   The default size is 4096. Minimum size allowed is 100.
;   Smaller buffer size may cause slowdown of the simulation.
;
;BNDBUFFERSIZE=1000
;
;
;   External control of minimum pipe length (m)
BRANCH_MIN_L=10
;
;   Two minimum depth criteria exist, relative and absolute depth.
;   For each link the minimum depth is set to the lowest of the
;   two criteria.
;
;   External control of the relative minimum water depth in per mille
;   in a link. Default value is 5 o/oo. Minimum accepted is 1 o/oo.
BRANCH_MIN_H_REL=5
;
;   External control of the minimum water depth in mm in a link
;   Default value is 5 mm. Minimum accepted is 1 mm.
BRANCH_MIN_H_ABS=5
;
;   External control of the minimum water depth in a manhole,
;   NEW_HMIN_NODE=1 gives the new improved formulation,
;   NEW_HMIN_NODE=0 gives the old formulation (MOUSE 3.23 and older
versions)
NEW_HMIN_NODE=1
;
;   External control whether to use the improved continuity equation or
not.
;   NEW_CONTINUITY=1 gives the new improved formulation
;   NEW_CONTINUITY=0 gives the formulation in MOUSE 3.23 and previous
versions.

```

```

NEW_CONTINUITY=1
;
; Switch for setting head loss and momentum computation back to
; the formulation used before MOUSE version 3.3.
; HEADLOSS_323=1 gives the old formulation used in version 3.23
; MOMENTUM_323=1 gives the old formulation used in version 3.23
; HEADLOSS_323=0 gives the new formulation. Default and recommended.
; MOMENTUM_323=0 gives the new formulation. Default and recommended.
HEADLOSS_323=0
MOMENTUM_323=0
;
; If a timestep less than 1.0 is stated here the engine will use that
; timestep as a fixed timestep throughtout the simulation.
;FIXED_DECIMAL_Timestep=0.2
;
; Matrix solver: 1: MOUSE Classic, 2: Iterative Method with assembler
vector math 3: Iterative method standard math
;SOLVER_METHOD=2
; Number of iterations allowed before shifting to the standard solving
procedure: No of nodes x ITERATION_FACTOR
;ITERATION_FACTOR=10
; Convergence threshold for the iterative procedure
;CONVERGENCE_LIMIT=1.0e-7
;
; The MOUSE Engine stops the simulation when dH or H conditions exceed
the range of
; the specified pump curves. This is the default if this entry is not
present or
; achieved by using PUMP_CURVE_RANGE_CHECK = 1
; It is possible to allow the simulation to continue by setting
; PUMP_CURVE_RANGE_CHECK = 0 (Available from Release 2009 SP3)
; In this case the pump discharge is set to the first and last value in
the table
; respectively. The period of operation outside the range of the pump
curve is
; reported to the Summary at the end of the simulation.
; It is recommended to inspect the summary carefully when
PUMP_CURVE_RANGE_CHECK = 0
; is applied.
PUMP_CURVE_RANGE_CHECK = 1
;
; Controlling the initialization of the water level in a pump pit.
; PUMPINITSTOPLEVEL=0 : The initial water level will be set at the
lowest pump START level (default)
; PUMPINITSTOPLEVEL=1 : The initial water level will be set at the
lowest pump STOP level
PUMPINITSTOPLEVEL=0
;
; Is possible to switch between different friction loss descriptions.
; External control of friction loss description in links (explicit or
implicit)
; imp_friction=1 activates implicit friction description
; imp_friction=0 (default) activates explicit friction description
('classic
; description)
; forward_cent = weighting coefficient, 1.0 results in a full forward
centering of the
; scheme
; kin_vis = kinematic viscosity of water [m2/s]. The default value is
for 15 deg. C.
; cw1,cw2,cw3,cw4=constants of Colebrook-White equation.
IMP_FRICTION=0
FORWARD_CENT=1.0
KIN_VIS=0.00000131
CW1=6.4

```

```

CW2=-2.45
CW3=3.3
CW4=1.0
;
; WRITESURFACEFLOODRESULTS=1 write flooding (flow rate to surface and
flood volume) results in XRF-file and HD summary
WRITESURFACEFLOODRESULTS=0
;
; WRITEINFLOWLIMITATIONSTORAGEVOLUME=1 write volume storage due to node
inflow restriction to results in XRF-file
WRITEINFLOWLIMITATIONSTORAGEVOLUME=0
;
; Generation of an extra *.XRF result file with:
; Flow velocity, Froude numbers, time step, volume in pipes and nodes
; Set PRINT_EXTRA_INFO=1 to generate the file.
PRINT_EXTRA_INFO=0
;
; All warnings can be switched off. The syntax is
; IGNORE_WARNING_XXX=1 where XXX is the number of the warning to be
; switched off.
;
IGNORE_WARNING_38=0
;
; OR all warnings can be suppressed before start of the simulation
; by setting PRINT_WARNINGS=0. Default setting is PRINT_WARNINGS=1
PRINT_WARNINGS=1
;
; Due to introduction of variable time step (MOUSE 3.3 and newer) the
former
; parabolic acceleration of pumps is changed to linear
; acceleration as the default approach.
; The parabolic acceleration can be switched on by setting
LINAR_PUMP_ACC=0
LINEAR_PUMP_ACC=1
;
; For MOUSE Long-Term (Job List) simulation, the result file post
; processing can be bypassed by setting the switch
LONGTERM_POSTPROCESS=0
; Default is LONGTERM_POSTPROCESS=1 which will always call post
processing.
LONGTERM_POSTPROCESS=1
;
; For writing a MOUSE650.out file which can be used as a base for ADP
files
; switch PRODUCE_MOUSE650OUT=1. Default is PRODUCE_MOUSE650OUT=0
PRODUCE_MOUSE650OUT=1
;
; Rising Main conversion factor. Default is 0.001
RM_CONV_FACTOR=0.001
;
; External control of the relative maximum pipe pressure for circular
pipes.
; The default value is 1000 times the pipe diameter.
;BRANCH_MAX_PRESSURE=1000.0
;
; Flood limit assigned for basins. I.e. the added height to the top
; of the basin geometry or ground level.
;FLOODLIMIT=25.0
;
; Reservoir height above ground, i.e. the distance from the manhole
groundlevel to where
; the expansion for ponding starts. 0.25 m below ground is the default
value.Note that when coupling
; MOUSE with MIKE 21 this is always increased to 50 m for the coupled
manholes only. The uncoupled manholes will follow the setting here.

```

```

; Make sure the FLOODLIMIT parameter
; does not conflict with the reservoir height.
;RESERVOIRHEIGHT=-0.25
;
; When a standard MOUSE model surcharges at the top of the manholes. The
Manhole is expanded to a 1000
; times the manhole diameter above ground. The RESERVOIRWIDTHFACTOR
allows for changing the default value.
;RESERVOIRWIDTHFACTOR=1000.0
;
; External control whether to use the corrected calculation of reservoir
above ground at surcharging.
; Additional precision is added to constants calculating expansion
coefficient, expansion height etc.
; NEW_RESERVOIRBASIN=1 gives the corrected formulation
; NEW_RESERVOIRBASIN=0 gives the formulation used in MOUSE/MIKE URBAN
2005 and previous versions.
NEW_RESERVOIRBASIN=1
;
;
; When modelling open channels the model will normally stop if flooding
occurs.
; Here it can be chosen globally to extrapolate all the cross-sections.
; CRS_EXTRAPOLATION=1 gives no extrapolation, i.e. equal to older MOUSE
versions (<2002)
; CRS_EXTRAPOLATION=2 gives an extrapolation according to the angle give
in:
; CRS_EXTRAPOLATION_ANGLE=90.0 is the angle between a horizontal level
and the extrapolated CRS
; CRS_EXTRAPOLATION_HEIGHT_FACTOR=3.0 is the maximum height of the
extrapolated CRS given
; as this factor times the maximum height of the CRS
;CRS_EXTRAPOLATION=1
;CRS_EXTRAPOLATION_ANGLE=90.0
;CRS_EXTRAPOLATION_HEIGHT_FACTOR=1.0
;
; During use of the LTS module - Joblist generation the model needs to
find out
; when it is raining and when it is dry weather. In order to avoid
influence from
; fluctuating rain gauges at low intensities it is possible to set the
VRAINZERO value.
; The default value is 0.00000001
;VRAINZERO=0.00000001
;
; Ensures that if it was raining e.g. 1.0 (Default) hours prior to a job
to be saved
; then the start of the event is moved to the start of this rain, and so
backwards
;MINHOURS BETWEENRAINS=1.0
;
; Switching INCLUDE_DTMINMAX_IN_JOBLIST on forces the joblist generation
in LTS to write
; the parameters for minimum and maximum timestep in each job
definition. By subsequently editing
; these parameters you can ensure that e.g. a particular difficult rain
event is modelled using smaller
; timesteps that needed for the rest of the jobs. (Default is 0 = turned
off).
;INCLUDE_DTMINMAX_IN_JOBLIST=0
;
; Switching IMPLICIT_PUMP on enables solution of pumps implicit
;IMPLICIT_PUMP=1
;

```



```

; It is possible to allow negative infiltration flow for pipes and
; canals (leakage)
; Set LeakageAllowed=1 if leakage should be allowed. Default is
LeakageAllowed=0
LeakageAllowed=0
; Leakage (negative infiltration) will be reduced to zero when the depth
; of water in
; is below the minimum depth. The linear reduction will start at
factor*minDepth
; The factor can be set by the entry "LeakageMinDepthFactor". Default
value is 2.0
LeakageMinDepthFactor=2.0
;
; When a non-return valve on a pipe is opened after being closed a 10
timestep startup period is used. The startup period
; can be specified by setting the NON_RETURN_VALVE_STARTUP_PERIOD.
; Default is off (= 10 timesteps). Unit is [s].
;
;NON_RETURN_VALVE_STARTUP_PERIOD = 5
;
; An alternative way of calculating the natural depth. This option can
be used when natural depth is the primary boundary
; in an inlet pipe where the water level fluctuates continuously.
; Set NATURAL_DEPTH_SLOPE_NEW=1 to use alternative depth calculation.
Default is NATURAL_DEPTH_SLOPE_NEW=0 (off).
;NATURAL_DEPTH_SLOPE_NEW=0
;
;----- M O U S E - M I K E 2 1 P A R A M E T E R S
-----
; MOUSE_MIKE21_LOGCONTENT=0 (Default) Gives initialisation information
in the log file but no timestep information.
; MOUSE MIKE21_LOGCONTENT=1 Gives initialisation information as well as
full timestep information.
;MOUSE_MIKE21_LOGCONTENT=0
;
; MOUSE_MIKE21_DVDT_MAX gives the maximal rate of change for the flow
in/out of manhole inlets
; Default value gives no reduction in the rate of change.
;MOUSE_MIKE21_DVDT_MAX=1
;
; MOUSE_MIKE21_ECOEF is an energy loss coefficient which is multiplied
on the flow through inlets in the coupling.
; A value of 1.0 (Default) gives NO loss
;MOUSE_MIKE21_ECOEF=1.0
;
; If there is a discrepancy between the ground level in MOUSE and MIKE
21 then the groundlevel in MOUSE
; is set to that of MIKE 21 if M21_AS_GROUNDLEVEL=1
;M21_AS_GROUNDLEVEL=1
;
; If the difference between the ground level in MOUSE and MIKE 21 is
bigger than
; MM21_ALLOWEDGROUNDLEVELDIFFERENCE then an error is issued. Unit is
[m].
;MM21_ALLOWEDGROUNDLEVELDIFFERENCE=0.1
;
; If the bottom level of an outlet in MOUSE is more than
MM21_ALLOWED_OUTLET_LEVELDIFFERENCE lower than the
; ground level in MIKE 21 then an error is issued. Unit is [m].
; Note that having large level difference can cause a mass balance
error.
MM21_ALLOWED_OUTLET_LEVELDIFFERENCE=0.1
;
; Sanity check for levels coming from MIKE 21. If water depths on the
surface is outside the range given by

```

```

; M21_MINDEPTH and M21_MAXDEPTH then minimum or maximum value is used
instead.
;M21_MINDEPTH=0.0
;M21_MAXDEPTH=100000.0
;
; The MM21_WARMUPPERIOD makes sure that no flow is exchanged through the
coupling until after the given
; number of timesteps. Default period is zero.
;MM21_WARMUPPERIOD=10
;
; The driving head for the computation is computed as the head between
MOUSE pressure level and the
; ground level in MIKE 21 plus MM21_HEAD_ABOVE_FLOODDEPTH. Default value
is 0.0.
;MM21_HEAD_ABOVE_FLOODDEPTH=0.002
;
;
; Silent MOUSE
;MM21_SILENT=1
;
;----- END OF M O U S E - M I K E 21 P A R A M E
T E R S -----
;
;
; Calculation of the energy loss coefficient for weirs, gates/orifices
for the km
; approach has been changed so that losses due to change in elevation
from weir,
; orifice/gate links has been removed
; It is possible to switch the application of the elevation loss On/Off
by the entry
; USE_ELEVATION_LOSS. The default is 'OFF'
;USE_ELEVATION_LOSS = OFF
;
; RTC debugging can be easier by dumping information on the selected
control functions
; for the different controlled devices
; PRINT_RTC_STATEMENTS=1 prints statement line numbers, the selected
control function
;                                     and relative time in hours of simulation time
; PRINT_RTC_STATEMENTS=2 prints statement line numbers, the selected
control function
;                                     and no time column
; PRINT_RTC_STATEMENTS=3 prints statement line numbers, the selected
control function
;                                     and the absolute time
;PRINT_RTC_STATEMENTS=3
;
; For optimization purpose the PERFORMANCE_MONITOR gives detailed
information on the
; time used for initialization.
;PERFORMANCE_MONITOR = ON
;
; The UND data validation can be disabled by using the UND_VALIDATION
keyword
; By doing so no validation will be performed on the input data thereby
risking crash if
; the setup contains invalid data. This should only be used to tweak
performance for
; models where the data is already validated.
;UND_VALIDATION = OFF
;
; The default slot width for sealed nodes for the water level moving
towards INF

```

```

; is 0.01 times the diameter of the manhole. This number can be adjusted
; here.
;NODE_SLOT_WIDTH = 0.01
;
; Improved way of handling low flow conditions (empty pipes). A tiny
; amount of water
; is needed in a numerical hydrodynamic model to keep the continuum of
; water in the
; model. When only the minimum of water level is required the roughness
; is increased
; and thereby decreasing the discharge. As soon as the water level rise
; above the
; minimum (e.g. 1mm) then the roughness is returned to the value
; specified by the user.
; Default is 0 (OFF)
;ALTERNATIVELOWFLOW = 1
;
; Velocity is by default written in the last Q grid point. This option
; enables writing
; velocity in the first grid point as well.
;CALC_VELOCITY_FIRSTQGP = 1
;
; This parameter determines how the engine considers the link length.
; LINK_LENGTH_TYPE_NO=1 Compute pipe length at center of manhole
; LINK_LENGTH_TYPE_NO=2 Compute pipe length at edge of manhole
;LINK_LENGTH_TYPE_NO = 1
;
; From Release 2007 of MIKE URBAN and MOUSE an improvement of
; pressurized flow conditions
; is available. The improvement involves modification of the standard
; cross section tables.
; The parameter CRSVERSION controls the applied version of the tables.
; The improved version is set to default (CRSVERSION=2). If the
; parameter is omitted
; or the parameter is set to CRSVERSION=1 the former tables will be
; applied.
CRSVERSION=2
;
; When specifying cross sections as a set of x,z,relative resistance
; values the user may
; select to use the effective area for calculating composite resistance.
; Note that this will
; impact the calculated velocities and the volume.
; Default is OFF
;UseEffectiveAreaRadius=ON

[TimeStepControl]
;
;WRITE_TIMESTEPCONDITIONS_SUMMARY=ON
;
;-----
; Parameters controlling the time step variation in the MOUSE HD
;-----
;
; QacceptLimitRel is used to ensure correct resolution of inflow
; hydrographs. The value specifies the maximal accepted relative
; deviation between the given and used inflow hydrograph.
; (Default = 0.10)
QacceptLimitRel=0.10
;
; QlowLimitM3s: The relative accept limit is not applied for inflows
; below 'QlowLimitM3s' m3/sec. (Absolute value)
; (Default = 0.010)

```

```

QlowLimitM3s=0.010
;
; WaterLevDiffMaxRel:
; Maximum relative change allowed for water depths between two time
; steps. (Relative value) Applied only for depths in standard pipes.
; (Default = 0.10)
WaterLevDiffMaxRel=0.10
;
; MaxVarCrossConstant:
; Is the maximal allowed variation of the cross section
; parameters: Area, Width and Hydraulic radius in one time
; step. (Relative value)
; (Default = 0.10)
MaxVarCrossConstant=0.10
;
; CrossLowDepthLimit:
; Parameter used for switching the checking of cross section parameters
OFF at
; low water depths. The parameter is the relative depth in the pipe.
; (Default = 0.04)
CrossLowDepthLimit=0.04
;
; Crosscheck:
; 0 means that cross-section variable are not checked
; 1 means that cross section variables check are
; based on MaxvarCrossConstant * (maximal
; value of the cross section)
; 2 means that cross section variables check are
; based on MaxvarCrossConstant * (actual
; value of the cross section)
; (Default = 1)
Crosscheck=1
;
; MaxPumpFlowVar:
; Limits the variation in the pump flow through one time
; step. (Relative value)
; (Default = 0.10)
MaxPumpFlowVar=0.1
;
; MaxCourant:
; Limits then variation of the courant number through one time
; step. (Absolute value)
; (Default = 20)
MaxCourant=20
;
[MouseTRAP]
;-----
;-----
; Section for control of MOUSE TRAP parameters
;-----
;-----
;
; As Sediment transport calculations often does not need to be carried
out
; every HD time step, it is possible here to set how often. I.e. a value
of 2
; means that the ST calculation will run every second time the HD
calculation
; runs. A value between 6-10 is recommended in the reference manual.
;ST_TimeStepMultiply=1
;ST_SaveStepMultiply=1
;
; External control of the transport mode for the fine PPC fraction.
; PPC_FINE_AD_TRANSPORT=1 transports the fine sediment by use of the
; AD module - PPC_FINE_AD_TRANSPORT=0 transports the fine sediment by

```

```

; use of the ST module
PPC_FINE_AD_TRANSPORT=1
;
; External control whether to print TRAP result items or not.
; PRINT_      =0 does not save the item in the result file.
PRINT_WEIR_AD=1
PRINT_PUMP_AD=1
PRINT_NODE_AD=1
PRINT_GRID_AD=1
PRINT_GRID_MASS_AD=1
PRINT_MASS_BALANCE_AD=1
PRINT_BASIN_ST=1
PRINT_WEIR_ST=1
PRINT_PUMP_ST=1
PRINT_SEDIMENT_TRANSPORT=1
PRINT_BED_LEVEL_ST=1
PRINT_MASS_BALANCE_ST=1
PRINT_BED_SHEAR_STRESS=1
PRINT_DIM_LESS_BED_SHEAR_STRESS=1
MIN_ST_DEPTH_REL=1
;
; DWFCONC allows the addition of a fixed level of sediment concentration
in mg/l
; to be added to the global fraction in the ST module.
;
;DWFCONC=200.0
;
;
; When simulating blends either every blend is calculated individually or
; the last blend is calculated based on subtraction. Default is all
blends
; are calculated.
; CalculateBlendFromSubtraction = 1
;
; Sediment transport
; Definition of minimum sediment deposit width, which will be used even
if no sediment is present.
; Use value > 0 to set initial conditions for ST. (Default is zero. May
result in no sediment transport).
; MIN_ST_WIDTH_AT_PCT_PIPEFILLING = 0
;
; Sediment transport
; Setting a maximum value for sediment transport to avoid infinite high
values at discontinuities. Unit is [mg/liter]. Default value is 10000
; MAXSEDIMENTTRANSPORT = 10000

```

## 1.1 Extended Explanations

### 1.1.1 Reporting flooding volumes and inflow limitation volumes to the Extra Result file (.XRF)

Although advanced methods exist for simulation surface flooding (e.g. dual drainage models or integrated 1D and 2D surface flooding models), in many cases it is useful to be able to get a rough estimate of surface flooding volume and flow rates at manholes due to system surcharging, or due to inflow limitations at the manhole. The dhiapp.ini file includes switches for writing the time-series of the two types of volume calculations to an 'extra results file' (.XRF).

To generate the Extra Results file you should apply the following entry in the 'dhiapp.ini' file. The default setting is not to produce the file.

```
; Generation of an extra *.XRF result file with:
; Flow velocity, Froude numbers, time step, volume in pipes and nodes
; Set PRINT_EXTRA_INFO=1 to generate the file.
PRINT_EXTRA_INFO=0
```

In addition to this the summary after the pipe flow simulation can provide information on this as well. This is achieved by the following entry in the 'dhiapp.ini' file:

```
WRITEINFLOWLIMITATIONSTORAGEVOLUME=1
```

An example of the output is shown in below:

<b>Nodes - Node inflow restriction</b>				
Node ID	Inflow restriction	Status at end of simulation	Volume stored at end of simulation	Max volume stored
	[m3/s]		[m3]	[m3]
10	0.150	Neutral	0.0	20.8
90	1.000	Neutral	0.0	0.0

Status: "Storing" = Inflow is being restricted  
 Status: "Releasing" = Stored volume is being released to node inflow  
 Status: "Neutral" = Inflow is not restricted

## Flooding volume and flow rates

For a condition where flooding occurs due to a system surcharges, the flow will be positive when water leaves the network during surcharge conditions, and flow will be negative when the surface flow returns to the network once the surcharge conditions have subsided. When surcharge occurs at 'Spilling' nodes, the flooded water will not return to the system and hence the flow can only be positive (or zero).

In addition information on the accumulated flooding volume can be printed to the simulation summary file. (.HTML).

The output of this information to the Extra Result file (.XRF) is controlled by the following entry in the 'dhiapp.ini' file. By default the feature is switched off (=0).

```
;
; WRITESURFACEFLOODRESULTS=1 write flooding (flow rate to surface and
; flood volume) results
; in XRF-file and HD summary
WRITESURFACEFLOODRESULTS=0
;
```

## Surface storage volumes due to inflow limitations at the manhole

For a condition where inflow to a manhole from a catchment is limited by the MaxInflow value, the Inflow Limitation volume represents the volume of water temporarily stored (held back) at the manhole due to the inflow limitation. The MaxInflow value is entered in the 'Nodes' dialog in MIKE URBAN Model Manager.

This volume storage does not influence the pressure level at the node. It is only a bookkeeping volume. There is no depth or geometry used in this computation.

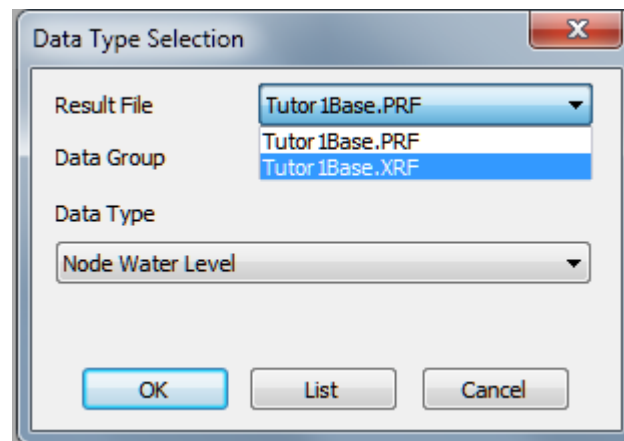
The output of this information to the Extra Result file (.XRF) is controlled by the following entry in the 'dhiapp.ini' file. By default the feature is switched off (=0).

```

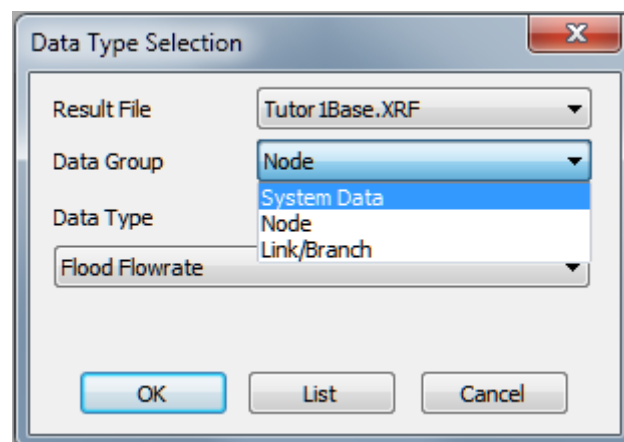
;
; WRITEINFLOWLIMITATIONSTORAGEVOLUME=1 write volume storage due to node inflow
; restriction to results in XRF-file
WRITEINFLOWLIMITATIONSTORAGEVOLUME=0
;

```

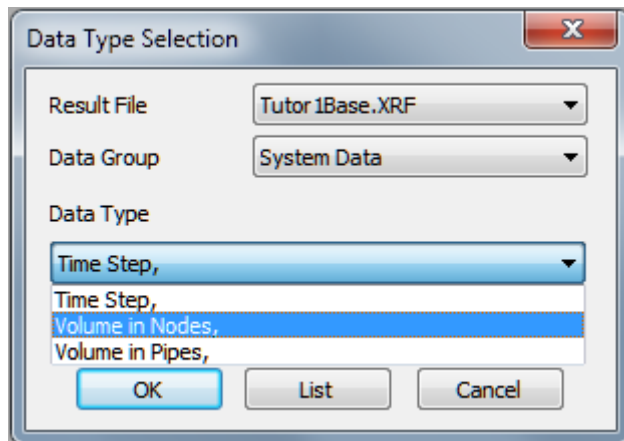
To present the results in MIKE VIEW you should first load the main result file (.PRF) and then use the <File>,<Add> menu and select the Extra Results file (.XRF). Open the Data Type Selection dialog and select the XRF file as the Result File.



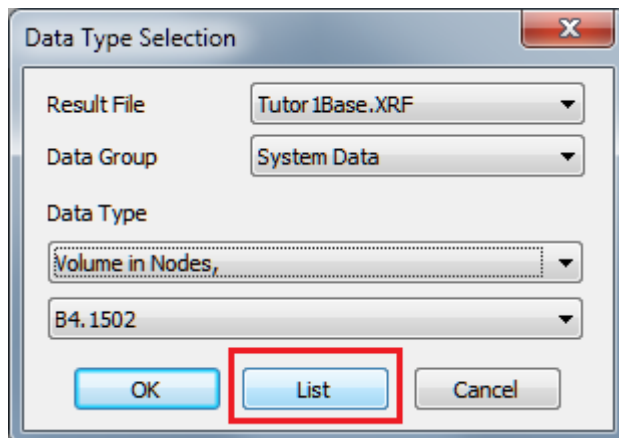
Then select 'System Data' in the Data Group option.



Then select 'Volume' as the Data Type.



Finally press the 'List' button. It is important to use the list in this case. The selection cannot be made from the Horizontal Plan view.

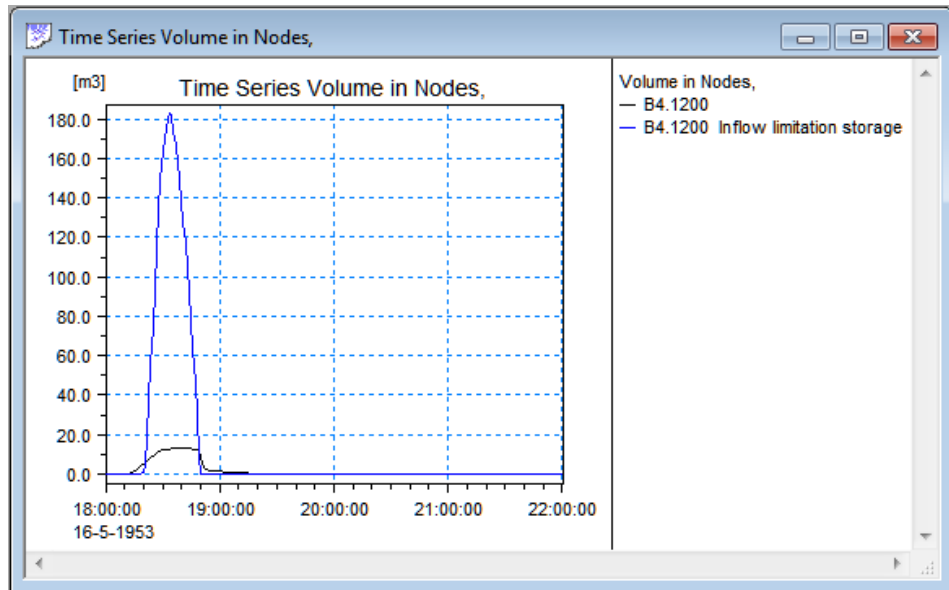


In the list you will first find all nodes for presentation of the time series of volume in the node. The nodes for which an inflow limitation has been defined will appear at the bottom of the list with their node ID and the text 'Inflow limitation storage'.

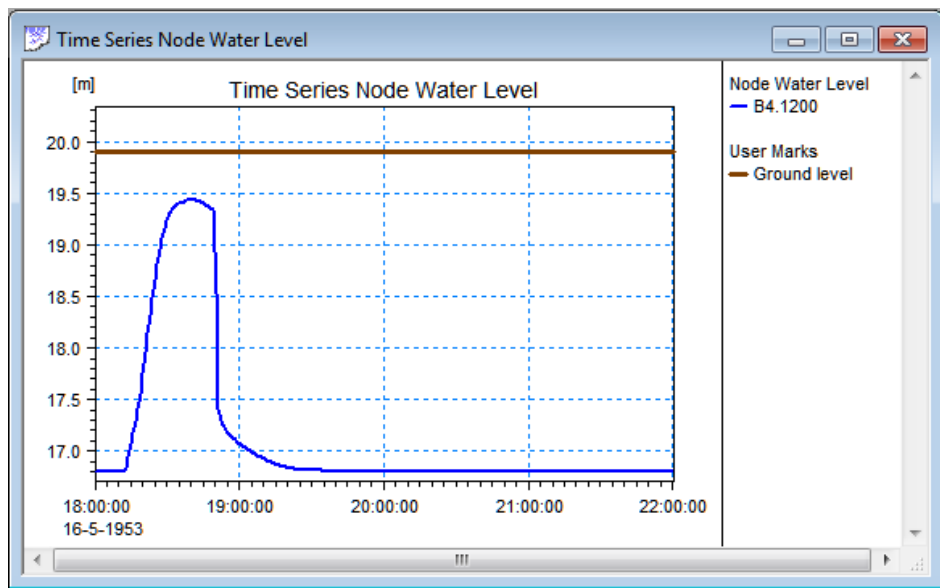
Item	Volume in Nodes,	Minimum	Maximum	Min.Time	Max.Time
1	B4.1502	0.004	75.805	16-5-1953 18:00:0	16-5-1953 18:36:5
2	B4.1501	0.004	28.078	16-5-1953 18:00:0	16-5-1953 18:33:3
3	B4.1500	0.009	59.627	16-5-1953 18:00:0	16-5-1953 18:34:3
4	B4.1491	0.007	22.507	16-5-1953 18:00:0	16-5-1953 18:31:0
5	B4.1490	0.015	5.022	16-5-1953 18:00:0	16-5-1953 18:34:3
6	B4.1485	0.025	7.029	16-5-1953 18:00:0	16-5-1953 18:37:5
7	B4.1320	0.025	2.083	16-5-1953 18:00:0	16-5-1953 18:32:3
8	B4.1480	0.050	8.908	16-5-1953 18:00:0	16-5-1953 18:51:1
9	B4.1200	0.017	12.945	16-5-1953 18:00:0	16-5-1953 18:39:5
10	B4.1310	0.025	1.817	16-5-1953 18:00:0	16-5-1953 18:26:4
11	B4.1300	0.025	2.294	16-5-1953 18:00:0	16-5-1953 18:26:4
12	A0.0327	0.000	0.000	16-5-1953 18:00:0	16-5-1953 18:00:0
13	B4.1520	0.008	50.491	16-5-1953 18:00:0	16-5-1953 18:28:4
14	B4.1510	0.050	59.848	16-5-1953 18:00:0	16-5-1953 18:28:4
15	B4.1200 Inflow limitation storage	0.000	182.979	16-5-1953 18:00:0	16-5-1953 18:33:3

In this example both the node volume and the Inflow Limitation Storage volume has been selected. This will show the following graph.

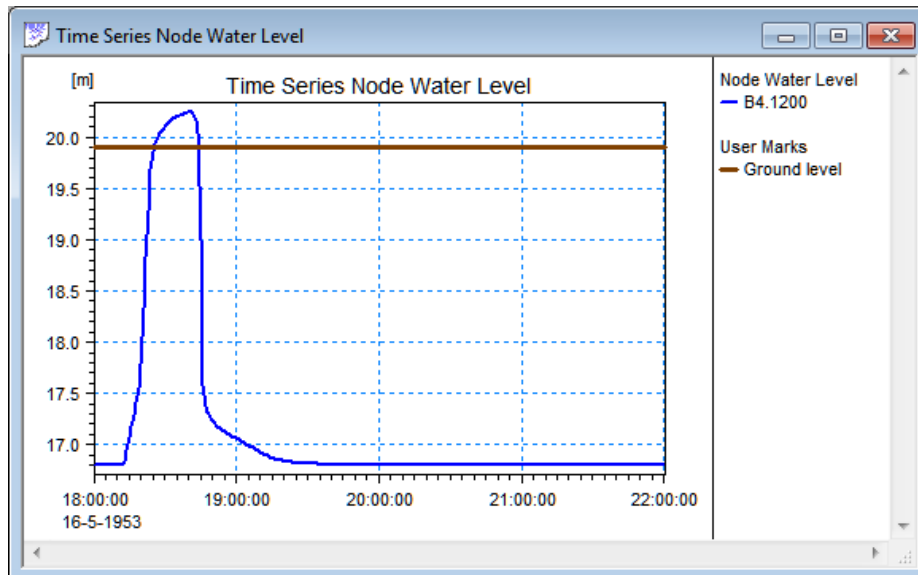
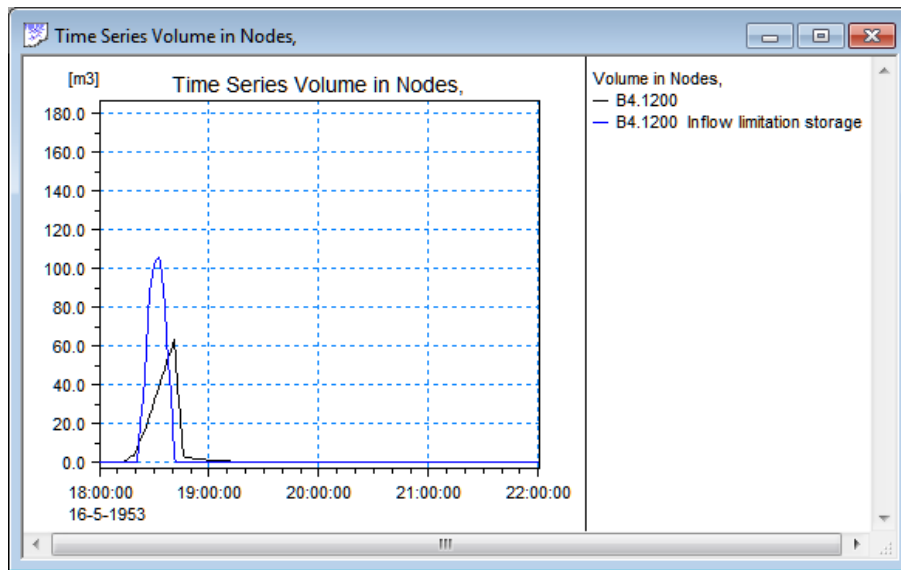




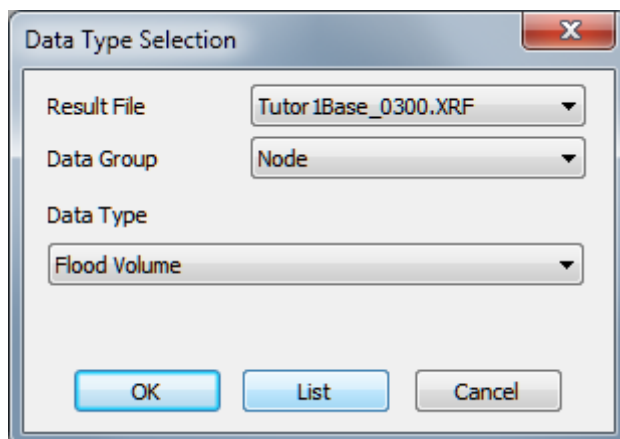
This is the result for an inflow limitation set to 0.300 [m3/sec] at node 'B4.1200' in the Tutor1 example distributed with MIKE URBAN. In this simulation the water level at the node 'B4.1200' stay below the ground level and no flooding due to surcharge occur.

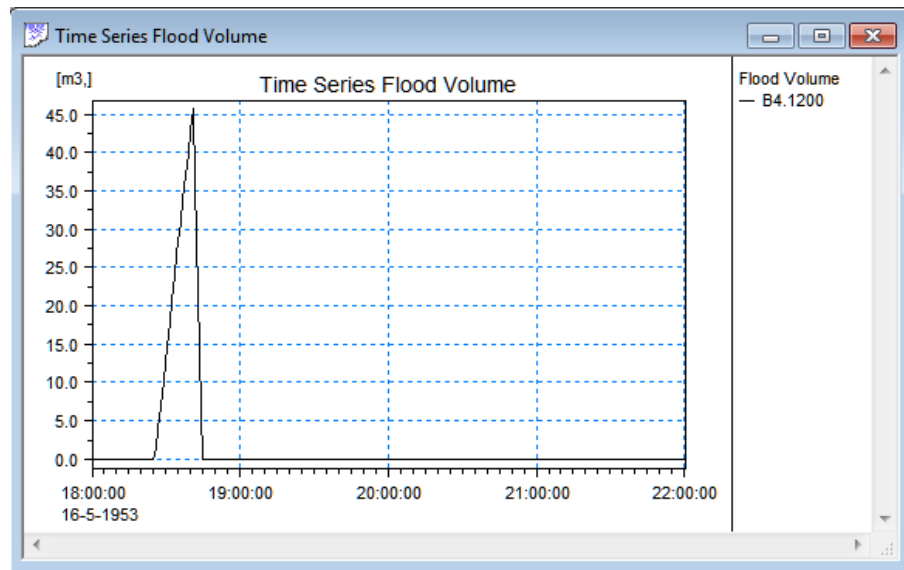


The effect of changing the inflow limitation to 0.400 [m3/sec] is show in the following charts. It is seen that this inflow limit cannot prevent flooding due to surcharge. The water level goes above ground level and the volume reported for the node includes volume on ground for a period during the simulation.

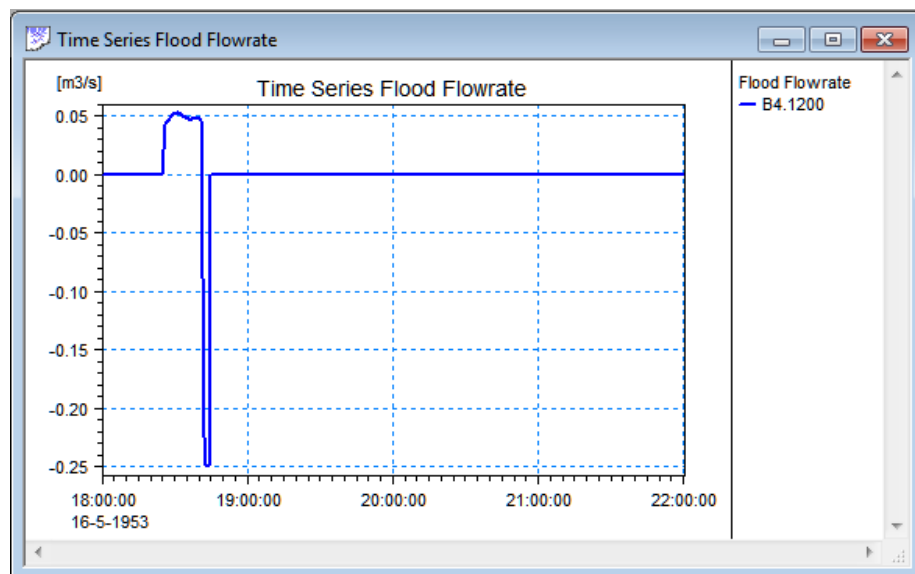


To show the surcharge flooding volume you can use this selection in MIKE VIEW:





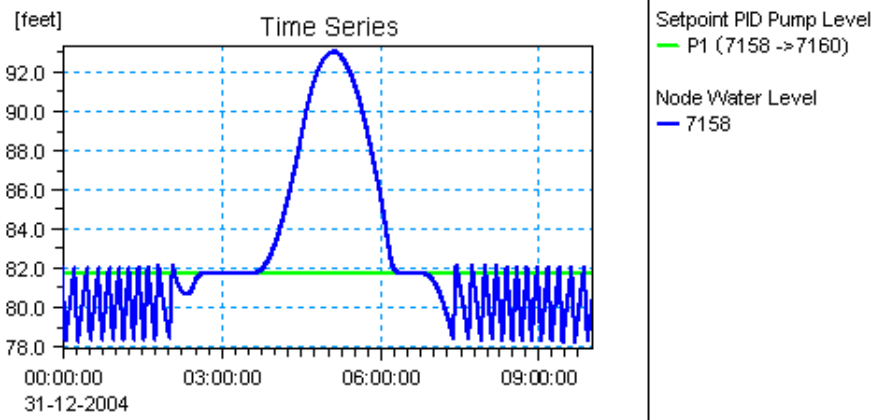
The flooding volume is then shown as a separate graph. This is the volume above ground level during due to surcharging during the simulation. Finally you can also show the flow rate for the surcharge flooding. Here positive values represent flow from the node to the ground (node expansion above ground), while negative values represents flow back into the node.



### 1.1.2 Variable speed pumps – output of operation

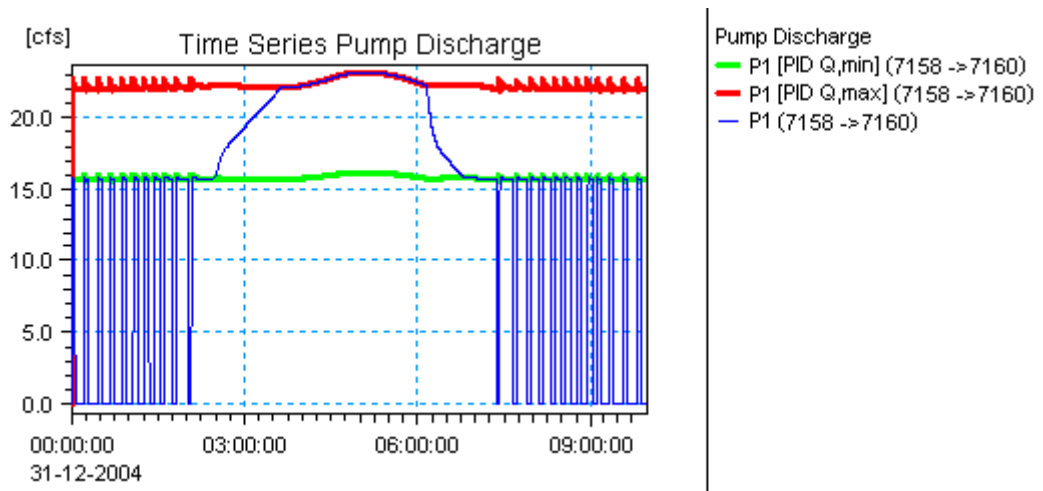
The capacity of the variable speed pumps will be controlled by the PID regulation algorithm to maintain the water level at the pump wet well according to the defined water level set-point value.

In some cases the water level may raise above the defined set-point as in the example shown in the figure below.



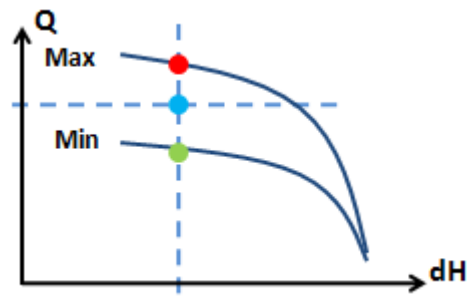
To provide more detailed information for this type of situation it is possible to output additional information to the standard result file. (Available from MOUSE Engine Release 2008)

In addition to the actual pump discharge the result file will also have time series showing the pump capacity from the two pump capacity curves defining the range of the pump “speed” variation. In the result presentation it will then be possible to combine these two time series with the actual pump discharge. The figure below shows how this can be presented in MIKE View.



From the figure above it is seen that the pump capacity for a period during the simulation is limited by the upper range capacity curve (red line). This is also the period where the water level raises above the defined set-point level.

The figure below indicates how the situation is when looking at the pump curves at time 03:00:00 in the above example.



The MOUSE Engine feature for output of the Q-values obtained from the 'RPMmax' and 'RPMmin' pump curves may be used for detailed analysis of the consequences of changing the capacity range or other conditions.

To apply this MOUSE Engine feature you should place a copy of the 'dhiapp.ini' file in the project work folder. The 'dhiapp.ini' file is located in the BIN folder of MIKE URBAN and MOUSE.

In the INI file you should insert the following lines in the [Mouse\_HD] section of the file:

```
WriteVarSpeedPumpExtraInfo = 1
```

```
WriteVarSpeedPumpExtraInfoTestVersion=1
```

**IMPORTANT NOTE:** With this information added the result file cannot be used as a hotstart file. The additional information is only intended for test purpose or for detailed analysis.

### 1.1.3 Grain size mobility

When running the sediment transport module in standard mode it will save results with bed shear stress and dimensionless bed shear stress. Instead of saving the dimensionless bed shear stress, the largest possible grain size which can be moved at a given time, can be saved. The computation is based on the relation between grain size and dimensionless bed shear stress

$$d = \frac{\tau}{\rho_w g (\rho_s - 1) \theta_{cr}} \quad (1.1)$$

where

$d$	grain size diameter [m]
$\tau$	bed shear stress [N/m <sup>2</sup> ]
$\rho_w$	density of water [kg/m <sup>3</sup> ]
$g$	gravitational acceleration [m/s <sup>2</sup> ]
$\rho_s$	density of sediment [kg/m <sup>3</sup> ]
$\theta_{cr}$	critical dimensionless bed shear stress [-]

The feature is turned on in the DHIapp.ini file by in the [MouseTRAP] section adding the line:

```
PRINT_GRAINSIZEMOBILITY_INSTEAD_OF_DL_BED_SHEAR=ON
```



## 2 The "\*.ADP" File

### 2.1 Introduction to \*.ADP

The \*.ADP file (ADditional Parameter file) serves as an additional model input possibility, which allows the user to access and control some program features and processes, which are not accessible through the MIKE URBAN/MOUSE menu system. Typically, the \*.ADP temporarily includes new MOUSE functions prior to their full implementation in the MOUSE user interface/MIKE URBAN, as well as some rarely used functions, which are of interest of only a few MOUSE users.

The \*.ADP is an ASCII file in the so-called PFS format, where information is organised in sections, and where entries are optional and may be given in an arbitrary order. Most of the parameter values in the \*.ADP file may be entered either in SI-units or in US-units.

The \*.ADP file is an optional input to a MOUSE computation. The selection of an \*.ADP file (if any) to be included into the current computation job is controlled through the MOUSE computational dialog, similarly as for any other optional input file.

#### 2.1.1 Detailed description of data sections in the \*.ADP file

The format of the '\*.ADP' file allows an optional input of various sections in arbitrary order. The syntax divides the file to sections starting with a line with [SECTIONNAME] and ending with a line containing "EndSect // SECTIONNAME". Sections may contain subsections. Within sections there can be defined any number of parameter lines starting with "KeyWord =" followed by the parameters associated to the key word. This means that the sequence of the keywords is not important (except for SYNTAX\_VERSION). Multiple specifications of the same type sections are also allowed. .

Note that the lines starting with '/' are comment lines. Also, all text beyond the '/' in a line is a comment text.

A detailed description of the each section in the '\*.ADP' file is given in the following paragraphs.

Some functionality is covered by the MIKE URBAN - MOUSE user interface while it is not covered by the MOUSE user interface. Please refer to specific sections for info on this. If not stated otherwise the functionality needs to be specified in the ADP-file for both MIKE URBAN and MOUSE.

## 2.2 General Features

### 2.2.1 Computing volume for a selected part of the CS network

It is possible to compute the actual volume of water contained in a part of the network as a time series output to the result file.

It is possible to define any number of volume calculations each covering various parts of the network. Each volume calculation will be identified by a unique “Volume ID” label which appears in the overview in the result presentation.

Each volume calculation can include any number of nodes and links. The specification is done in the ADP (Additional Parameter) file.

For easy configuration the volume calculation definitions can apply references to ‘selection files’. For MIKE URBAN the selection files will be ‘.mus’ files and for MOUSE the selection files will be ‘.nse’ for nodes selections and ‘.lse’ for links selections.

When the definition of the volume calculations is loaded there will be issued error messages if specified nodes or links does not exist in the network model. If a node or link is specified more than once within the same volume calculation definition there will be issued a warning. The duplicated node or link will only be included once in the volume calculation.

The most simple example will be one volume calculation based on a selection file (\*.mus file) provided from MIKE URBAN.

```
[VOLUME_SELECTIONS]

[VOLUME_CALCULATION]
  VolumeID = 'Volume Series 1'
  SelectionFile = 'Volume1.mus'
EndSect // VOLUME_CALCULATION

EndSect // VOLUME_SELECTIONS
```

This example will result in a time series in the result file from the MOUSE Engine with the identification “Volume Series 1”. The volume will include the water contained in the nodes and links included in the selection file from “Volume1.mus” created from the MIKE URBAN Model Manager.

There must be one and only one KeyWord “VolumeID” in each [VOLUME\_CALCULATION] section. There can be any number of KeyWord “SelectionFile” in the [VOLUME\_CALCULATION] sections.

If more “SelectionFile” appears then the information from each file will be merged.

There can be any number of the subsections [VOLUME\_CALCULATION]. Each of these will result in a separate time series of volume data in the result file.

In addition to using selection files the nodes and links can also be specified directly by using the keywords “NodeID” and “LinkID”. The next example shows how the options can be combined.



```
[VOLUME_SELECTIONS]

[VOLUME_CALCULATION]
  VolumeID = 'Volume 1'
  SelectionFile = 'Nodes.nse'
  SelectionFile = 'Links.LSE'
  NodeID = 'N10'
  LinkID = 'L10'
EndSect //

[VOLUME_CALCULATION]
  VolumeID = 'Volume 2'
  NodeID = 'N20'
  NodeID = 'N21'
  LinkID = 'L20'
  LinkID = 'L21'
EndSect //

EndSect //
```

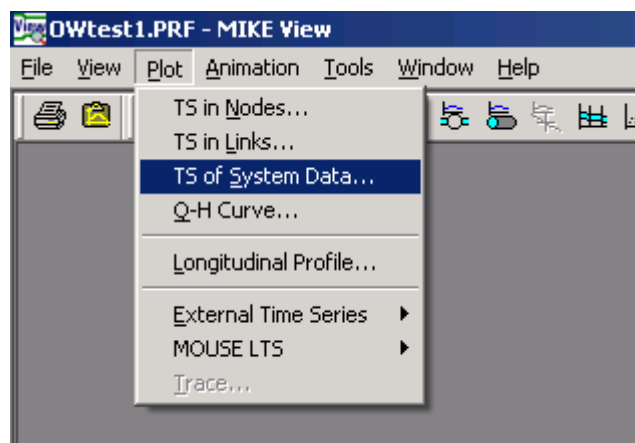
The example includes in this case two volume calculations, 'Volume 1' and 'Volume 2'. The first uses selection files from MOUSE (.nse and .lse) in combination with individual specification for node 'N10' and link 'L10'.

The second volume calculation 'Volume 2' has only individual specifications of nodes and links.

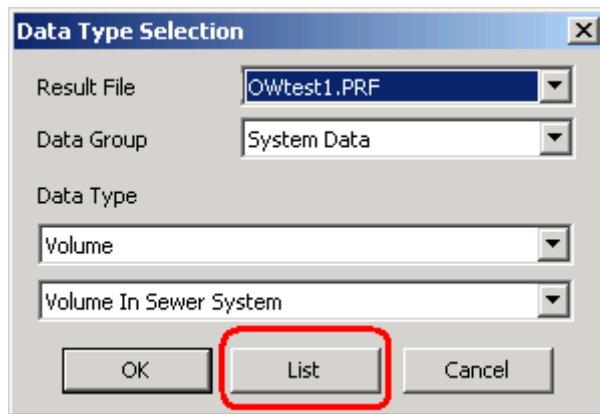
### Presenting the computed volume time series in MIKE View

The volume time series calculated according the specifications in the ADP file will be output to the standard result file from the MOUSE HD simulation. The '.PRF' file.

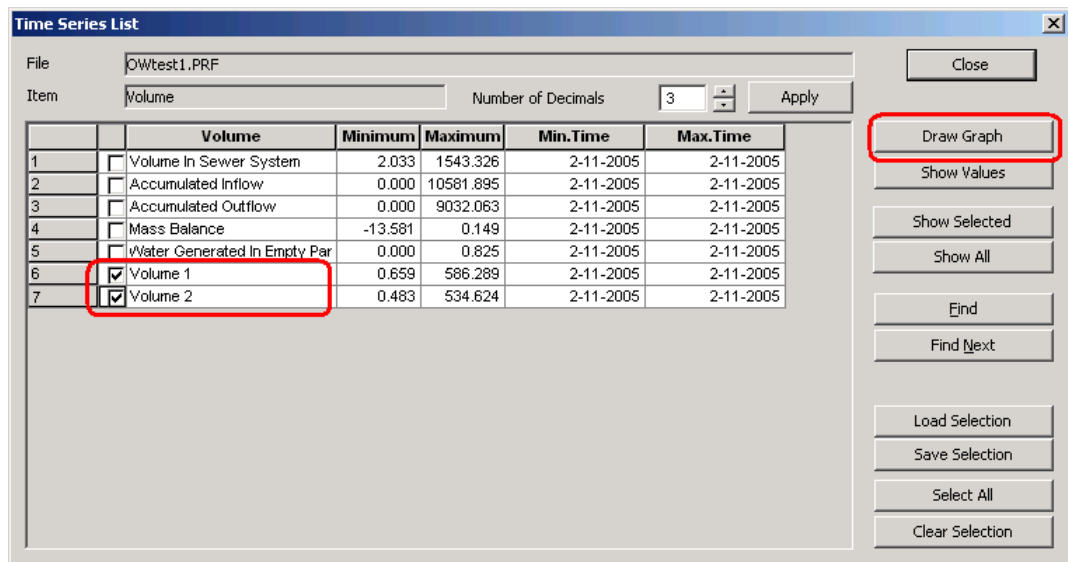
When the result file is loaded into MIKE View it is possible to present the volume time series by opening the <Plot> menu and selecting <TS of System Data ...>.



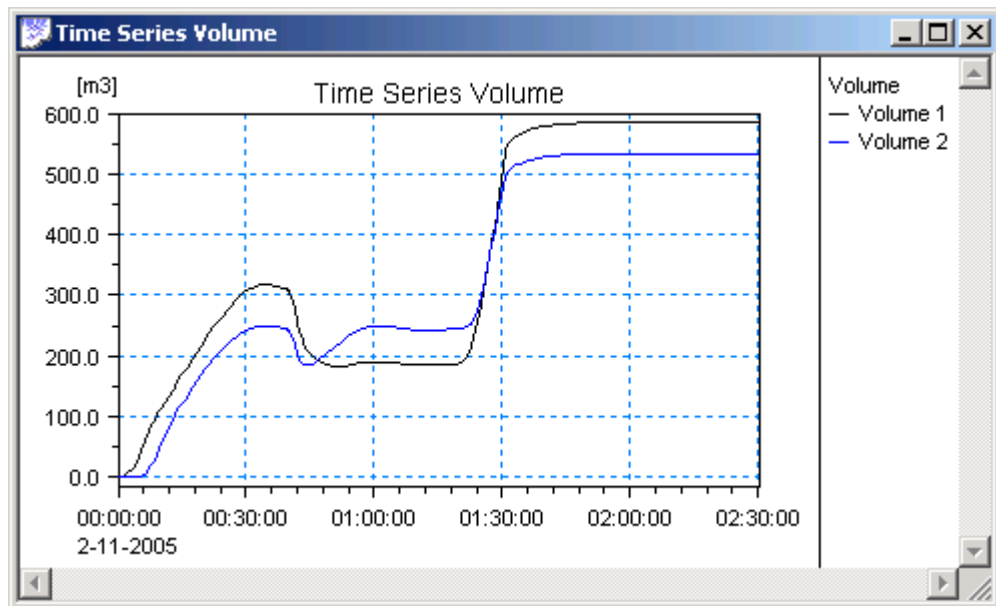
This will open the Data Type Selection dialog.



Continue to the list of volume time series by pressing the <List> button.



In the Time Series List of 'Volume' the two specified volume time series 'Volume 1' and 'Volume 2' will appear. Selecting the time series and pressing the <Draw Graph> button will open a graph window with the result of the computed volumes.



### 2.2.2 Initial water level – set from any node

As a standard feature the MOUSE Engine will initialize the water level in the network according to the water level specified at outlet nodes (through the boundary system). The water level will be set as horizontal back water from the outlet reaching all parts of the network below this level with free access for the water from the outlet node.

For some network models this feature is not sufficient. For various reasons some models may not have outlet nodes at all. This will be the case when all water is pumped out from the network.

To handle this and similar cases the MOUSE Engine has an additional feature for setting the initial water level from any node in the network. The feature is available through input via the ADP file (Additional Parameters file). The textbox below shows a simple example of how the feature is used from the ADP file.

```
[INITIAL_WATER_LEVEL]

SYNTAX_VERSION = 1.0
UNIT_TYPE      = 1      // 1 = SI units  2 = US units

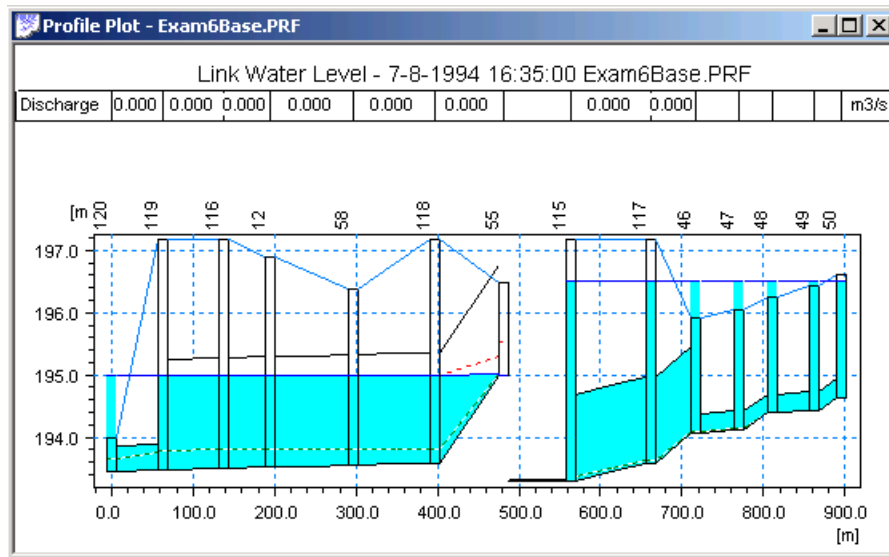
InitialWaterLevel = '46', 196.50
InitialWaterLevel = '58', 195.00

EndSect // INITIAL_WATER_LEVEL
```

If the Node ID is specified as " (i.e. empty string), then the specified water level is applied over the entire model. When a Node ID is specified, then the initial horizontal water level is set from this node. This then works according to the same rules which apply for specifying water level at an outlet. This means that the water level specified is not passed through pumps or through weirs with crest level above the specified water level.

Note that this option can be applied to set different initial water levels in separated sections of the model.

With UNIT\_TYPE = 1 the specified water level is [m], with UNIT\_TYPE = 2 the specified water level is [ft]



### 2.2.3 Variable headloss coefficients

The structure of this section is identical to tabular data in the UND file

```
[MOUSE_TABULAR_DATA]
UNIT_TYPE = 1
[DataSet]
  DataSetID = '<NodeID>'
  TypeNo = i
  Data = A1, B1
  Data = A2, B2
  Data = An, Bn
EndSect // DataSet
EndSect // MOUSE_TABULAR_DATA
```

Unit\_Type indicates either SI units (1) or US units (2). TypeNo indicates which type of relation is issued.

- 1 = Level - Km / Contraction HLC / Total HLC
- 2 = Discharge - Km / Contraction HLC / Total HLC
- 3 = Velocity - Km / Contraction HLC / Total HLC

Data is issued where A is monotonously increasing.

## 2.3 MOUSE LTS - Long Term Simulations - Features

### 2.3.1 LTS - output of the Stop Criterion flow

In the configuration of Long Term Simulations (LTS) you must apply a Stop Criterion for defining when the detailed simulation should end after each rain event and jump in time to the start of the next rain event.

The setting of the Stop Criterion has in some projects caused some trouble because it is difficult to get an overview of the actual flow which the Stop Criterion is evaluated against. This MOUSE Engine feature solves the issue by an option for generating an output file with the flow time series during the Job List Generation.

The produced time series will be the sum of Storm Runoff provided from the Runoff Simulation result file (CRF file) and the Dry Weather Flow and all other network loads which will contribute to the total inflow to the network during the simulation.

With MOUSE Engine Release 2008 you can get the time series written to a text file with data in the format which can be imported directly in MIKE View as an external time series.

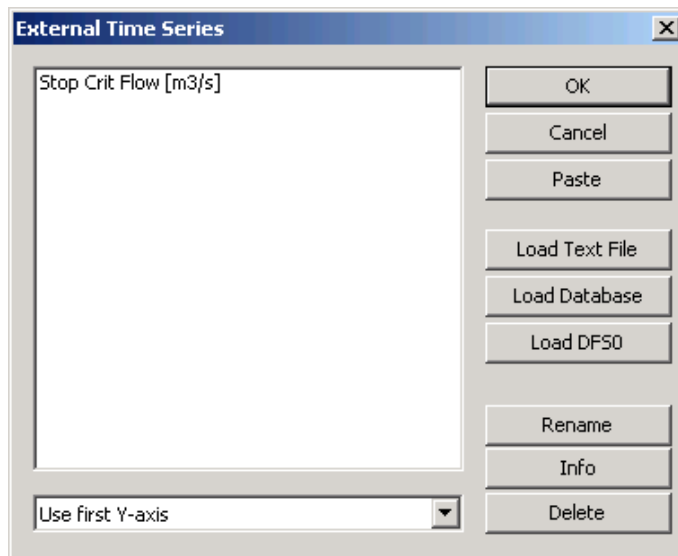
```

Stop Crit Flow [m3/s]
2006-01-02 00:15:00    0.0085
2006-01-02 00:30:00    0.0072
2006-01-02 00:45:00    0.0062
2006-01-02 01:00:00    0.0049
2006-01-02 01:15:00    0.0058
2006-01-02 01:30:00    0.0036
2006-01-02 01:45:00    0.0044
2006-01-02 02:00:00    0.0043
2006-01-02 02:15:00    0.0037
  
```

To make the MOUSE Engine produce the output file you should just place an empty file with the file name 'STOPCRITFLOW.TXT' in the project folder. The reported flow will be in [m3] or [cft/s] according to the unit setting for the actual simulation.

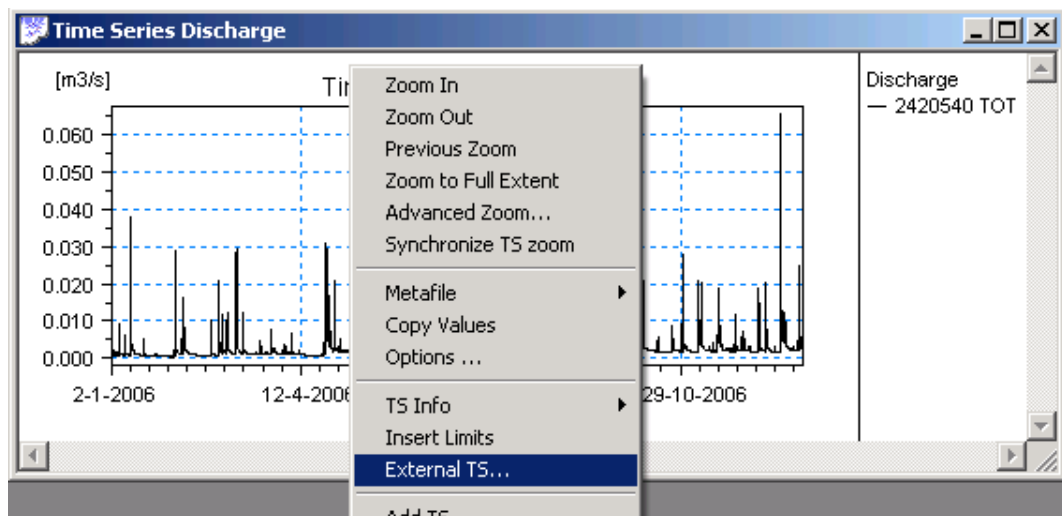
To present the time series of the total inflow you can e.g. open the result file from the Runoff Simulation in MIKE View and then add the text file as an external time series.

In MIKE View select: <Plot>, <External Time Series>, <Edit...>, then select <Load Text File>. Open the file named STOPCRITFLOW.TXT, confirm using the first line as the name for the time series. The External Time Series dialog will now appear as in the figure below.



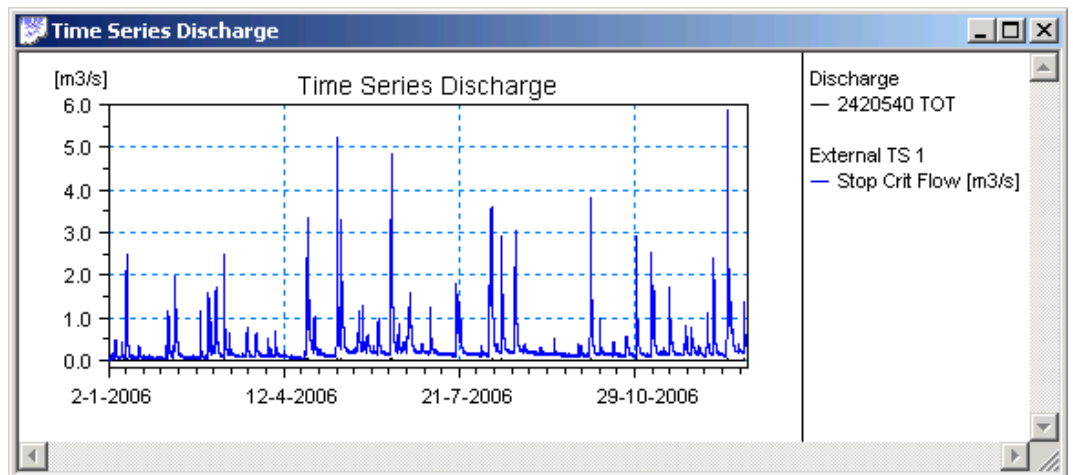
Press the <OK> button to close the dialog. The select and plot any of the hydrographs from the Runoff Result file.

Make a right-click in the graph to open the local menu and select External TS...



This will again open the External Time Series dialog. Highlight the “Stop Crit Flow” and press the buttons <Paste> and then <OK>.

The Stop Criterion Flow time series is now presented in the graph.



### 2.3.2 LTS - additional output from Job List generation

Before making MOUSE Long Term Simulations you must first generate a job list which will be used during the simulation to control the switching between periods with detailed simulation and jumping between rain events.

The Job List is generated based on defined criteria for starting the simulations. Setting the criteria is a task which needs to be considered carefully. It will be a balance between having sufficient number of rain events included for the actual analysis and avoiding too long simulation time.

It is possible to get more information about how the defined start criteria are affecting the resulting Job List.

The Job List is output to a text file with the file extension '.MJL' (Mouse Job List).

The file lists the generated Jobs with their start and end date and time and eventually the hotstart file and time step to apply.

From Release 2008 the file will have additional information written as comment lines for the individual jobs as well as a summary at the end of the file. The summary shows for how many jobs the defined start criteria were applied.

```
[SIMULATION_EVENT]
Simulation_start = '1955-08-28 11:24:00'
Simulation_end = '1955-08-28 12:58:00'

// Job number :    28
// Duration    :    0 01:34:00
// Start criterion ID : JL_Start_2
EndSect // SIMULATION_EVENT
```

### Example - Additional information output to each job specification

```
// Number of jobs      :    115
// Start criterion ID : JL_Start_1 was used in :    38 jobs.
// Start criterion ID : JL_Start_2 was used in :    25 jobs.
// Start criterion ID : JL_Start_3 was used in :    52 jobs.
// Total duration     :     20 17:50:00
```

### Example – Summary

## 2.3.3 LTS - DWF insertion between events

When running a discontinuous calculation (MOUSE LTS) it can be desirable to save reasonable flow values in periods where the dynamic model is not run, i.e. between events. In MOUSE this is not supported by a graphical interface, but it is possible to enter information on where to insert flow values by writing directly in the \*.ADP file.

The flow saved between events consists of dry weather flow (DWF), inflow from boundary conditions and inflow from the rainfall runoff, if there is any (CRF). The catchment to include when composing the flow needs to be specified as seen below

#### **IMPORTANT:**

First a regular RSF file must be created as the flow is inserted in the user specified result file.

The section to be added in the ADP file must look like the section shown below:

```
[MOUSE_LTS_INSERTFLOW]
// Insert simple flow values between events: InsertFlow = true
InsertFlow = true
SaveStep_HrMiSec = 0, 20, 0
[Locations] // UpNode, Dwnode, NodeSeletiontype, Node/nse/'
  BRANCH = 'B4.1520', 'B4.1510', 1, ''
  BRANCH = 'B4.1200', 'B4.1485', 2, 'Nodes.nse'
  BRANCH = 'B4.1480', 'A0.0327', 3, 'B4.1480'
EndSect // Locations
EndSect // MOUSE_LTS_INSERTFLOW
```

The insert flow option turns the feature on and off. SaveStep\_HrMiSec makes it possible to choose another interval for saving dry weather results than the one chosen in the RSF specification (which applies for saving results during rain events).

The Node selection type can be either 1, 2 or 3, where 1 means that all catchments will be connected to this branch, 2 means that the following node selection file will contain the nodes which will be connected to the branch and 3 means that only the individual node stated will be connected to the branch.



## 2.4 Link to Groundwater Model MIKE SHE

```
[MOUSE_COUPLING]
  SYNTAX_VERSION = 1
  UNIT_TYPE = 1
  CALLER = 'MSHE'
  COUPLINGMMSHE= 'ID', 'LinkType', 'OLConductivity
  ', 'OLConductivityExponent', 'SzLeakageCoeff '
  EndSect //EFOR_PARAMETERS
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [MOUSE\_COUPLING] section. The parameter value 1.0 indicates the syntax version 2004.

The present version of MOUSE MIKE SHE link requires that data is given in SI units and the UNIT\_TYPE keyword must be defined to be 1.

The CALLER keyword must be defined to 'MSHE'.

The COUPLINGMMSHE keyword defines a MOUSE – MSHE LINK, and there is not limitation in the number of links.

The <ID> parameter is the MOUSE ID of the link (the ID of the Node or the Link)

The <LinkType> (integer) is a code of the Link (1 means the two models are linked at a MOUSE Node, 2 means the two models are linked at MOUSE branch). A MOUSE node link is coupled to the overland component of MIKE SHE, a open branch to the overland competent and the ground water component while a closed branch (a pipe) only is linked to the ground water level of MIKE SHE. MIKE SHE calculates location of the links on the basis of the geographical location of MOUSE elements (x, y, z (invert and top elevation)), please refer to the MIKE SHE reference manual for details.

The <OLConductivity> is a parameter of the link attached to t the overland flow, please refer to the MSHE reference manual for details. In a MOUSE node the unit of the parameter [m3-OLExchangeExpo/s], while the unit at a MOUSE branch is [m2-OLExchangeExpo/s]

The parameter <OLConductivityExponent> is the exponent of the MIKE SHE overland flow link computation.

The parameter <SzLeakageCoeff> is the Leakage coefficient (i.e. conductance per unit wetted perimeter per unit length) applied in the calculation of groundwater exchange.

## 2.5 Structures

### 2.5.1 Control of pump behaviour & output from these

In this section a number of keywords related to the control of pump behaviour and the output from pumps will be specified.

## Operation of pumps outside the pump curve range

In practical modelling you may experience situations where the hydraulic conditions only for a very short period result in dH values outside the tabulated range. This will cause the MOUSE Engine to stop the simulation with an error message.

A possible solution to this problem may be to add values to the pump curve table and hereby let the simulation pass through the period where the dH values exceed the original range of the table.

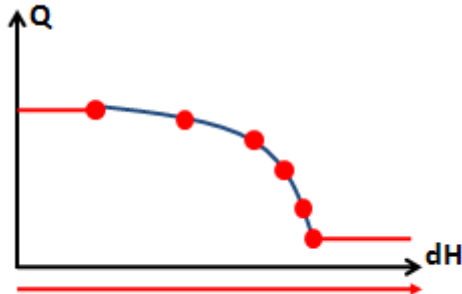
However, the disadvantage of this approach is that you may run simulations where the pump for some periods is forced to operate in unrealistic conditions. In this case the simulation work may result in misleading results and conclusions.

## Extending the pump curves

The MOUSE Engine offers a feature for allowing the pumps to operate outside the defined pump curve table and report this when it happens. In this case you can keep the pump curve with the limited range as defined by the manufacturer.

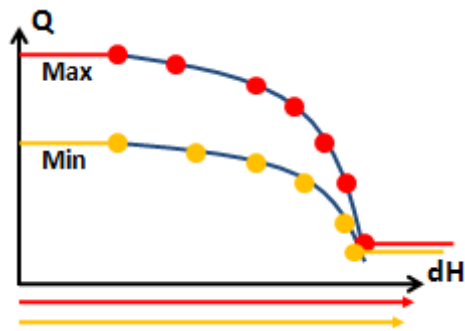
The feature is controlled through an entry in the 'dhiapp.ini' file.

By allowing the pump to operate outside the pump curve range the simulation will always continue, also when the hydraulic conditions results in dH values outside the table. In practice the pump curve will in this case be as shown in the figure below.



When the pump is allowed to operate outside the defined pump curve it is valuable information to know, if this unrealistic situation actually happen during the MOUSE Engine simulation. Therefore the summary provided at the end of the simulation has been extended within the 'Pumps - Discharge' section. Here you will find columns with information about the total time of operation of the pump, and how long time the operation was at dH values below or above the tabulated range found in the pump curve table. (For information on variable speed pumps, please refer to the Collection System User Guide).

It is possible also for variable speed pumps to let the MOUSE Engine continue the simulation when the conditions exceed the pump curves. This sets the pump curves to be as shown in the figure below.



When the pump operation is allowed to exceed the pump curve range during the simulations then the summary report available at the end of the simulation will have information about the actual operation. The information is found in the "Pmps – Discharge" section.

MOUSE HD Computation Engine v2009 Release Version (0.0.0.3174)

MOUSE Pipe Flow Simulation — Status Report — Dynamic Wave

Index of summary

[File Overview](#)  
[Time Overview](#)  
[Agent Overview](#)  
[Time Step Parameters](#)  
[Constants Balance](#)  
[Boundary Connections](#)

**Pumps - Discharge**

	Minimum	Maximum	Flow Accumulated	Time - Minimum	Time - Maximum	Pump starts	Dry stops (s)	Speed	Operation total	Operation above dH or H range of RPMmax curve	Operation below dH or H range of RPMmax curve	Operation above dH or H range of RPMmin curve	Operation below dH or H range of RPMmin curve
	[cfs]	[cfs]	[cfs]			[Count]	[Count]		[Hr:Min:Sec]	[Hr:Min:Sec]	[Hr:Min:Sec]	[Hr:Min:Sec]	[Hr:Min:Sec]
Balboa_Lead	0.000	18.624	510991.0	2004-12-31 00:00:00	2004-12-31 02:50:20	17	0	Variable	7:58:50	0:00:00	7:58:50	0:00:00	7:58:50

(1) : Pump stops due to dry pump well.

The table contains columns showing if the pump is constant speed pump or variable speed pump. The actual time of operation is shown in the next column. After this follows columns with information about duration of operation below or above the defined range of the 'RPMmax' and 'RPMmin' pump curves.

The contents of these columns may as an example be like in the table below. In this case the summary shows that the pump during the complete operation has been running at conditions exceeding the range of both pump curves.

Table 2.1 Pumps - Discharge

<b>Speed</b>	Variable
<b>Operation total</b> [Hr:Min:Sec]	7:58:50
<b>Operation above dH or H range of RPMmax curve</b> [Hr:Min:Sec]	0:00:00
<b>Operation below dH or H range of RPMmax curve</b> [Hr:Min:Sec]	7:58:50
<b>Operation above dH or H range of RPMmin curve</b> [Hr:Min:Sec]	0:00:00
<b>Operation below dH or H range of RPMmin curve</b> [Hr:Min:Sec]	7:58:50

In this case the simulation result should be investigated further and if may be considered to select an alternative pump.

For constant speed pumps only the 'RPMmax' columns will be used.

### Switching the pump curve range check on/off

The feature for allowing pumps to operate outside the range of the pump curve is controlled by an entry in the 'dhiapp.ini' file.

In the section [MouseHD] the keyword 'PUMP\_CURVE\_RANGE\_CHECK' is used for controlling the range check during the simulation. Setting the value for this keyword to 0 will switch the check off and allow the range to be exceeded. Setting the value to 1 will set the check on. This is also the default setting by the MOUSE Engine used if the keyword is not present in the 'dhiapp.ini' file (see below).

```

; The MOUSE Engine stops the simulation when dH or H conditions exceed
; the range of
; the specified pump curves.
; This is the default setting achieved by using PUMP_CURVE_RANGE_CHECK =
1
; It is possible to allow the simulation to continue by setting
; PUMP_CURVE_RANGE_CHECK = 0
; In this case the pump discharge is set to the first og last value in
the table
; respectively. The period of operation outside the range of the pump
curve is
; reported to the Summary at the end of the simulation.
; It is recommended to inspect the summary carefully when
PUMP_CURVE_RANGE_CHECK = 0
; is applied.
PUMP_CURVE_RANGE_CHECK = 1
    
```

### 2.5.2 Radial Gates

```

[RADIAL_GATE]
// GATE = 'GateID', Trunnion, Radius, TranBottom, TranDepth,
RadialGateOverflow, GateCoef, GateExp, GateHeight
EndSect
    
```

The Radial Gate Section lets the user modify an existing gate to be a radial gate with the associated parameters

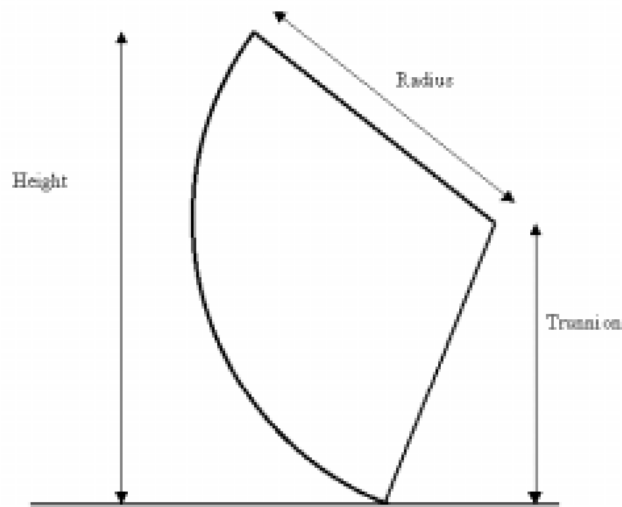


Figure 2.1 Radial Gate

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [RADIAL\_GATE] section. The parameter value 1.0 indicates the syntax version 2003. No earlier versions are available.

Trunnion:	Height above sill of centre of gate circle, see Figure 2.1.
Radius:	Radius of gate, see Figure 2.1.
Tran. Bottom:	Parameter used to define the transition zone between free flow and sub-merged flow.
Tran. Depth:	Parameter used to define the transition zone between free flow and sub-merged flow.
RadialGateOverflow:	Indicate that RadialGateOverflow should be used. To enable RadialGateOverflow should be 1 to disable 2. If enabled all the following parameters must be issued.
Gate Coeff:	Coefficient used when calculating the flow above the gate.
Gate Exp:	Coefficient used when calculating the flow above the gate.
Gate Height:	Height above sill of the overflow crest of the gate when the gate is closed.

### 2.5.3 Explicit weirs

The computation speed of large models can be significant. In some cases it can be relevant to sacrifice some precision to gain speed. The MOUSE software solves the St. Venant equations using an implicit scheme (please see MOUSE Reference manual (pipe flow)). This solution technique ensures the highest possible precision. In cases where the pipe network is looped many places it is very time consuming to compute the solution.

The explicit weir functionality changes the weirs to be computed explicitly, cutting loops where weirs are located in two. This reduces the total computational time considerably. In one case the explicit weirs helped increasing the speed of the model with a factor of 18.

Please note that a speed improvement is only possible for looped networks and the weirs need to be internal weirs, i.e. weirs that discharges to another location in the network. In cases where it is not certain that the model will benefit from explicit weirs with regard to speed it is however very easy to test applying the input data definition given below. Changing otherwise implicit weirs to explicit ones may cause some instability and loss of precision, so it is a good idea to compare the new model results with results from the original model in at least a few of the weir locations.

The explicit weirs can be defined in the ADP file as seen below.

```
[EXPLICIT_WEIRS]
  //CalcMethodNo: 1=Implicit, 2=Explicit
  LineHeader = 'WeirID', 'CalcMethodNo', 'Centering', 'QFactorMax'
  Line = 'IntWeir1', 2, 0.4, 1.0
  Line = 'IntWeir2', 2, 0.5, 1.0
EndSect // EXPLICIT_WEIRS
```

The 'CalcMethodNo' can switch the weir between implicit and explicit formulation.

The 'Centering' factor must be between 0 and 1. If it is e.g 0.4 it means that the weir flow at timestep n+1 is computed as 0.4 times the flow computed at the last timestep and (1-0.4) times the newly computed value. The centering helps to smoothen out instabilities and a default value of 0.5 should be a reasonable value.

The 'QFactorMax' is only used when the weir is submerged. It gives an upper limit on the  $H/\Delta H$  factor seen in equation 2-16 to 2-18 in the MOUSE Reference manual (pipe flow) in the section concerning submerged weirs.

## 2.6 Control

### 2.6.1 PID Sampling Period

```
[PID_SAMPLING_PERIOD]
  SYNTAX_VERSION = 1.0
  STRUCTURE = '<Pump/Weir/OrificeID>', TypeNo, SamplingPeriod
EndSect
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [PID\_SAMPLING\_PERIOD] section. The parameter value 1.0 indicates the syntax version 2003. No earlier versions are available.

The STRUCTURE keyword includes the name of a PID controlled device, either a pump, weir or gate. The type number indicate the type where 1 is pump, 2 is weir and 3 is orifice. The sampling period is in seconds.

## 2.6.2 Control of Coupled Pumps

```
[PUMPQCOUPLINGS]
  SYNTAX_VERSION = 1.0
  PUMPQCOUPLING = '<Pump1>', '<Pump2>', ..., '<Pumpn>'
EndSect // PUMPQCOUPLINGS
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [PUMPQCOUPLINGS] section. The parameter value 1.0 indicates the syntax version 2002. No earlier versions are available.

The PUMPQCOUPLINGS keyword includes groups of PID controlled coupled pumps. Each group consists of two or more pumps, where the highest flow dictates the flow for all the other pumps in the group.

The <Pumpn> parameter specifies a pump within the group.

## 2.7 MOUSE Pipe Design

Input data for the Pipe Design Module is specified in the Additional Parameters File (ADP). It is possible to use the wizard for setting up the ADP-file (MIKE URBAN: Tools|Pipe Design Wizard – for further please refer to the Collection System User Guide) or the file can be created manually in e.g. Notepad. In order to activate the design module the simulation is started in the usual way after specification of the ADP file name on the Start Simulation Dialog, but an additional dialog for viewing and editing the design data will be available in parallel to the MOUSE Simulation Launcher.

The format of the DESIGN section in the ADP file is as follows:

```
[DESIGN]
  SYNTAX_VERSION = 1.0
                    UNIT_TYPE = 1
  COMMERCIALDIAMETERFILE = 'MyOwnDiameters.txt'
                    DESIGNRESULTFILE = 'MyOwnResults.txt'
  MANNINGITEM = 'Manning', 1, 1, 2, 'SelectedPipes.lse', '', 50, 200, -0.05,
  1.0
EndSect // DESIGN
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [DESIGN] section. The parameter value 1.0 indicates the syntax version 2003. No earlier versions are available. Unit\_Type indicates either SI units (1) or US units (2).

Keyword	Parameter	Description
COMMERCIAL DIAMETERFILE	"DIAMETERFILE"	File with standard diameters. Not mandatory. One diameter in each line specified in mm/inches and in ascending order.
DESIGNRESULTFILE	"RESULTFILE"	File containing the history of the individual diameters. If present at simulation start a continuance of the previous iterations is assumed.
DESIGN CONSTANTS	DC1-DC6	Parameters for internal use. Not mandatory.
MANNINGITEM	DESIGNID	DesignID is used to identify the actual design group, but has no influence on the simulations.
	DESIGNTYPE	1 – Fixed Invert levels for pipes 2 – Fixed Top levels for pipes
	DESIGNCRITERIATYPE	1 – Critical Levels (specified at node dialog) 2 – Filling Degree (value specified as parameter 10 FILLINGDEGREE)
	DESIGNGROUPTYPE	1 – Group. All pipes are resized in parallel based on criteria in one node (value specified as parameter 6 "UPSTREAMNODE") 2 – Individual Pipes. Each pipe is designed based on criteria in the immediate upstream node.
	ELEMENTS	LSE-file specifying the group of links to be designed
	UPSTREAMNODE	Upstream node where design criteria is evaluated. Only used for DESIGNGROUPTYPE = 1
	MIN	Minimum value of the Manning number M. Note that extreme low values can cause numerical problems during the simulation. Its recommended to use values relatively close to the original value and run a few more iterations instead.
	MAX	Maximum value of the Manning number M. Note - se above.
	LOWERLIMIT	Limit for evaluating the design criteria. Specified relative the design criteria.
	FILLINGDEGREE	Filling degree. Only used for DESIGNCRITERIATYPE=2



## 2.8 Biological Processes and Pollution Transport

### 2.8.1 Hydrogen sulphide build-up

```
[H2S_PARAMETERS]
SYNTAX_VERSION = 1
UNIT_TYPE = 1 // 1 = SI, 2 = US (only SI supported!!)
HS_Computation = true
SaveHSRate = true, 'HSrates.txt'
//HS_Equations:
//1: Thistlethwayte,
//2: Boon & Lister
//3: Pomeroy & Perkhurst
//4: Nielsen & Hvidtved-Jacobsen
//5: WATS model - Tanaka and Hvidtved-Jacobsen
HS_Equation = 4
//WWDegradability (Waste Water Degradability):
//1.5 - Domestic WW (Denmark),
//3.0 - Food Industry WW,
//6.0 - Very degradable
WWDegradability = 6.0
//Correct for full running pipes ra := - HS.Co * fFactor
CorrectForFull = true, 0.1
//Component names as they appear under TRAP | AD | Components
//If PH or H2S = '' then H2S is not calculated
//If Z = '' then Z-formula is not calculated
//The rest of the components are required
DO_Name = 'DO'
Temperature_Name = 'Temperature'
XOD_Name = 'BOD'
SO4_Name = 'SO4'
PH_Name = 'PH'
HS_Name = 'HS'
THS_Name = 'TotalHS'
H2S_Name = 'H2S'
H2Srelease_Name = 'H2Srelease'
Z_Name = 'Z'
VFA_Name = ''
OUR_Name = ''
//OURmodel = MeasuredOUR, mv_H, k_half, q_m, E_A, E_An, n_fe,
//q_fe, K_fe, Y_Hw, Y_Hf, alpha_w, alpha_f, continued...
//alpha_T, K_Sw, K_Sf, K_X1, K_X2, K_h1, K_h2, continued...
//nReacOrder, UseiDisTotRatio, iDisTot, continued...
//K_ns, K_H2S, p_s, K_O, K_Os, P_atm, P_VS, S_S, continued...
//S_A, S_F, X_Bw, X_Bf, X_S1, X_S2, S_Afac, continued...
//S_Ffac, X_Bwfac, X_Bffac, X_S1fac, X_S2fac1, X_S2fac2
OURmodel = . . .

//-----
// H2SReleaseOutputOption = 0 : Output as [mg/(m*s)] to TRF file
//                               mg per meter per sec, for each grid
point
// H2SReleaseOutputOption = 1 : Output as accumulated value [mg/m]
// Note: "1" must only be applied when "PRINT_GRID_MASS_AD=0"
// (default)
//       in the 'dhiapp.ini' file. Otherwise accumulation may be
violated
//       in some grid points.
//
// H2SReleaseOutputOption = 1 is used as default if nothing is specified
//-----
H2SReleaseOutputOption = 1
EndSect // H2S_PARAMETERS
```

## The parameters description

Scientific and technical reference information for the hydrogen sulphide build-up model is available in the relevant reference documentation. The information presented here refers to the input data requirements and formats.

The <Syntax\_Version> (integer) must be '2', denoting the version 2001/2002. No earlier versions are available.

The <Unit\_Type> (integer) must be '1' (i.e. SI). US units are not supported in version 2002.

The <HS\_Computation> (boolean) switches the HS model ON/OFF.

The <SaveHSrate> (boolean, string) switches ON/OFF writing of an output file with the time series of  $r_a$ . The string states the file name (including path). If only a file name is specified (i.e. no file path), the file will be saved in the working directory.

The <HS\_Equation> (integer) controls which of the equations will be used for the calculation, according to the following:

- |   |   |                                   |
|---|---|-----------------------------------|
| 1 | = | Thistlethwayte,                   |
| 2 | = | Boon & Lister,                    |
| 3 | = | Pomeroy & Perkhurst,              |
| 4 | = | Nielsen & Hvidtved-Jacobsen.      |
| 5 | = | WATS: Tanaka & Hvidtved-Jacobsen. |

The <WWDegradability> (double) defines the degradability of the wastewater. Ranges approximately between 1 to 6, where 6 is highly degradable.

The <CorrectForFull> (boolean/double). The first parameter switches the correction of  $r_a$  ON/OFF, when the pipe is pressurised. The second parameter is the calibration factor applied for the correction.

The following parameters (string) establish the link to the water quality components defined in the AD model ("TRAP|AD|Components"):

<DO_Name>	Name of the oxygen component.
<Temperature_Name>	Name of the temperature component.
<XOD_Name>	Name of the BOD or COD component. The actual meaning depends on which equation has been chosen.
<SO4_Name>	Name of the SO4 component.
<PH_Name>	Name of the pH component.
<HS_Name>	Name of the HS component.
<THS_Name>	Name of the Total HS component.
<H2S_Name>	Name of the H2S component.
<H2Srelease_Name>	Name of the H2S release component.
<Z_Name>	Name of the "Z component" in the AD model. This "component" serves for saving the results of the Z-formula in the result file (*.TRF).
<VFA_Name>	Name of the Volatile Fatty Acid component
<OUR_Name>	Name of the Oxygen Uptake Rate component

If <PH\_Name> and <H2S\_Name> are specified as empty strings ( ' ' ), then H<sub>2</sub>S is not calculated.

If <Z\_Name> is specified as an empty string ( ' ' ), then the risk factor Z is not calculated.

Specification of all remaining components is compulsory.

For explanation of the model parameters needed for the WATS model, please refer to the MOUSE TRAP Reference manual (WQ).

<H2SReleaseOutputOption> Option for controlling the parameters output to the TRF result file. Can be output as rate or as accumulated.  
(Available from Release 2014 SP2).

## 2.8.2 Inverted siphons

The sediment transport models are originally developed for mild slope rivers. This means that siphons are not modelled reliably with these formulae. Literature (Fredsoe, 1992) has proposed to adjust the critical bed shear stress by

$$\theta_c = \theta_{co} \cos \gamma \left[ 1 - \frac{\tan \gamma}{\tan(\varphi_s)} \right] \quad (2.1)$$

where

$\theta_{co}$  is the original dimensionless bed shear stress

$\gamma$  is the slope [°]

$\varphi_s$  is the static friction angle (angle of repose) for the sediment. App. 26° for sand and 33° for gravel

The correction of the critical bed shear stress has been implemented into the model for marked steep pipes. Marking steep pipes are done through a section in the ADP-file, which could look like below for a case with three steep pipes in the model:

```
[STEEP_PIPE_TRANSPORT]
SYNTAX_VERSION = 1.0
LINK = 'Spipe_1'
LINK = 'Spipe_2'
LINK = 'Spipe_3'
EndSect // STEEP_PIPE_TRANSPORT
```

## 2.8.3 Sediment traps

The MOUSE ST software can now deal effectively with sediment traps. The currently implemented basin removal function is aimed at sediments entering large retention basins. Research has shown that investigated sediment traps has effectively been able to catch approximately all of the sediments travelling as bed load given that they are emptied often enough. The bed load is taken out of the flow at these locations given that there is storage volume left in the trap. The bed load has been calculated based on the relatively simple Meyer-Peter and Müller bed load formula:

$$\left. \begin{aligned} \Phi_B &= 8(\theta - \theta_c)^{3/2} \\ \Phi_B &= \frac{q_B}{\sqrt{(\rho_s - 1)g d^3}} \end{aligned} \right\} \Rightarrow q_B = 8(\theta - \theta_c)^{3/2} \cdot \sqrt{(\rho_s - 1)g d^3} \quad (2.2)$$

where

$q_B$  is the bed load transport per unit width [ $m^3/s / m$ ]

The sediment traps can be specified in the MOUSE ADP file. The three sediment traps have been specified in the ADP file as shown below:

```
[SEDIMENT_TRAPS]
SYNTAX_VERSION = 1
UNIT_TYPE = 1
//HEADER = BasinID, MethodNo, n, TotalVolume [m3],
//InitialFreeVolume [m3],CleaningPeriod [days],
//LastCleaningDate [dd-mm-yyyy hh:mm:ss]
//MethodNo: 1=Efficiency formula, 2=Bed load (Peter-Meyer)
//trapping, 3= Efficiency formula with trapping
//n: Efficiency factor (see reference manual) for method 2
//it also serves as reduction factor multiplied on the bed
//load.
//CleaningPeriod: Days between cleaning, eg.7.0 means weekly
//LastCleaningDate: 0 means at start of simulation but
//initial free volume remainsas stated. The date-time format
//must be the same as the setting in the PC's system.
SEDIMENTTRAP = 'TRAP No1', 2, 1.0, 22.0, 22.00, 14.0, '01-02-1994
00:00:00'
SEDIMENTTRAP = 'TRAP No2', 2, 1.0, 10.0, 10.00, 14.0, '01-02-1994
00:00:00'
EndSect // SEDIMENT_TRAPS
```

As seen it is possible to specify a reduced efficiency through the n-factor. It is also possible to include the cleaning procedures in the model as long as the grit pits are cleaned with a regular interval.

## 2.8.4 Fixed sediment depth in individual conduits

Simulations can be conducted with specified fixed sediment depth in individual conduits. The geometry of the cross-section with sediment depth specified is modified by cutting-off the cross-section part below the specified depth, and the hydraulic conveyance is recalculated accordingly.

This feature is not available through the graphical interface, but it is provided through the ADP – file only.

The syntax is the following:

```
[CONSTANT_SEDIMENT_DEPTHS]
SED_DEPTH = 'Link_MUID',Sediment_depth_value
...
...
...
EndSect // CONSTANT_SEDIMENT_DEPTHS
```

## 2.9 Sections only Relevant for Discontinued MOUSE Package

### 2.9.1 Links (section only relevant for MOUSE)

```
[ADDITIONAL_LINK_PARAMETERS]
SYNTAX_VERSION = 1.0 // 1.0 = VERSION 2001/2002
UNIT_TYPE = 1 // 1 = SI, 2 = US
MANNING_TYPE = 1 // 1 = Manning M, 2 = Manning n=1/M
LINK_MANVAR = '<LinkID>', <ManBot>, [<ManTop>, <ManExp>]
LINK_LGRIDSLOT = '<LinkID>', [<Length>], [<Number Grid>],
                                     ' [<Slot Width>]
EndSect // ADDITIONAL_LINK_PARAMETERS
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [ADDITIONAL\_LINK\_PARAMETERS] section. If omitted (or the parameter value set to 0.0), the model will assume the syntax as in version 2000B or earlier. The parameter value 1.0 indicates the syntax version 2001 or 2002.

The UNIT\_TYPE keyword, which controls the units (SI or US), is optional. If omitted, use of SI units is assumed. If present, the UNIT\_TYPE keyword must be found only once in a [ADDITIONAL\_LINK\_PARAMETERS] section. This has impact on the parameters specified after the keyword LINK\_LGRIDSLOT (m or ft).

The MANNING\_TYPE keyword indicates whether the Manning parameters after the LINK\_MANVAR keyword are specified as Manning M or Manning  $n=1/M$ . If omitted, use of Manning M is assumed. The MANNING\_TYPE keyword must be found only once in a [ADDITIONAL\_LINK\_PARAMETERS] section, immediately after the [SYNTAX\_VERSION] and [UNIT\_TYPE].

The LINK\_MANVAR keyword is used for entering constant or depth varying Manning values in the following syntax:

```
LINK_MANVAR = '<LinkID>', <ManBot>, [<ManTop>, <ManExp>]
```

In earlier syntax versions, instead of '<LinkID>' the node names associated with the upstream and downstream end of the link ('<UpNode>', '<DwNode>') were used to locate the link in the model. Note: If more links exist in parallel the parameters will be applied for all these links.

<ManBot> is the Manning number at the bottom of the pipe or for the complete depth if the optional parameters <ManTop>, <ManExp> are omitted.

<ManTop> is the Manning number at the top of the pipe.

<ManExp> is the coefficient controlling the Manning number variation with the depth. See the MOUSE Tech. Reference for details about the Manning number variation.

The LINK\_LGRIDSLOT keyword is used for assigning individual values for pipe length, the number of computational grid points, the grid spacing and the slot width of the Preissmann Slot at the top of the pipes, using the following syntax:

```
LINK_LGRIDSLOT = <LinkID>', [<Length>], [<Number grid>], [<Slot Width>]
```

In earlier syntax versions, instead of '<LinkID>' the node names associated with the upstream and downstream end of the link ('<UpNode>', '<DwNode>') were used to locate the link in the model. Note: If more links exist in parallel the parameters will be applied for all these links.

All other parameters are optional. Place commas alone to indicate position of used parameters.

<Length> is the length of the pipe [m]. Use this parameter if the length different than the length computed automatically based on the X-Y co-ordinates of the adjacent nodes or the length specified in the link dialog is wanted.

<Number grid> is the number of computational grid points to apply for this link. The value should be an odd number  $\geq 3$ . If an even number is found then MOUSE will add one to the specified value.

<Slot Width> is the width of the Preissmann slot (relative to the conduit width).

Notice that setting of these parameters may create numerical problems in critical locations of a MOUSE model. Also, inappropriate settings of these parameters may severely violate the results of the MOUSE computations, without apparent stability problem. Please refer to the MOUSE Reference Manual before using these parameters. All of the parameters are set automatically by MOUSE if they are not found here.

## 2.9.2 Colebrook – White (section only relevant for MOUSE)

```
[COLEBROOKWHITE_FRICTION]
SYNTAX_VERSION = 1.0 // 1.0 = VERSION 2001/2002
UNIT_TYPE = 1 // 1 = SI, 2 = US
CW_LINK = '<LinkID>', <Roughness>
...
EndSect //COLEBROOKWHITE_FRICTION
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [COLEBROOKWHITE\_FRICTION] section. If omitted (or the parameter value set to 0.0), the model will assume the syntax as in version 2000B or earlier. The parameter value 1.0 indicates the syntax version 2001 or 2002.

The UNIT\_TYPE keyword, which controls the units (SI or US), is optional. If omitted, use of SI units is assumed. If present, the UNIT\_TYPE keyword must be found only once in a [COLEBROOKWHITE\_FRICTION] section. This has impact on the parameter <Roughness>.

The parameter <Roughness> specifies the equivalent pipe wall roughness (m or ft), used in Colebrook-White formula.

### 2.9.3 Control of PID Minimum Capacity Curve (section only relevant for MOUSE)

```
[PUMPMINCAPACITY]
SYNTAX_VERSION    = 1.0

[PUMPMINCAPACITYTABLE]
PUMPID = '<PumpPID>'
H = <H1>, <H2>, ... , <Hn>
Q = <Q1>, <Q2>, ... , <Qn>
EndSect // PUMPMINCAPACITYTABLE
EndSect // PUMPMINCAPACITY
```

The SYNTAX\_VERSION keyword is optional and, if present, must be located on the top line in a [PUMPMINCAPACITY] section. The parameter value 1.0 indicates the syntax version 2002. No earlier versions are available.

The [PUMPMINCAPACITY] section is used for specifying a lower capacity curve for PID controlled pumps. A standard PID controlled pump will, in order to obtain the specified setpoint, operate between 0 and the pump capacity curve. Including this section in the ADP-file will only allow the pump to operate between the lower capacity curve (specified here) and the capacity curve specified through the interface.

The [PUMPMINCAPACITYTABLE] section specifies the capacity curve for each pump.

The PUMPID keyword specifies the relevant pump. The pump must be specified as PID controlled.

The H and Q keywords specify the points in the capacity curve.

### 2.9.4 Additional DWF definition

In DWF it is possible to offset a repetitive profile in time. This value is entered in hours.

An example of the format of the Additional DWF Definition section in the ADP file is as follows:

```
[DWF_ADDITIONAL_DEFINITION]
G_DWF_LINE = '', 0.0, 1
G_DWF_LINE = '', 0.0, 2
G_DWF_LINE = '', 0.0, 3
G_DWF_LINE = '', 0.0, 4
L_DWF_LINE = 'Catchments1.cse', 0, 2
L_DWF_LINE = 'Nodes1.nse', 0, 2
I_DWF_LINE = 'B4.1200', 0, 1
I_DWF_LINE = 'B4.1502_4_4', 0, 2
EndSect // DWF_Additional_Definition
```

The keywords are G\_DWF\_LINE for general, L\_DWF\_LINE for list and I\_DWF\_LINE for individual. The first parameter is the specific name associated with the DWF item. For general, there is no name, for list it is the name of the catchment/node selection file and for individual it is the node/catchment id.

The second parameter is the value that the time should be offsetted with. This value is in hours. The last parameter is an integer indicating: 1: Residential, 2: Commercial, 3:Base Flow or 4:Specific.

## 2.10 Pump Emergency Storage Estimator (section only relevant for MOUSE)

The pump emergency storage is defined as the storage volume that exists within a sewer network between the alarm level and the overflow level before wastewater is discharged to the environment.

A schematic of components of the pump station and network that would contribute to emergency storage is provided below.

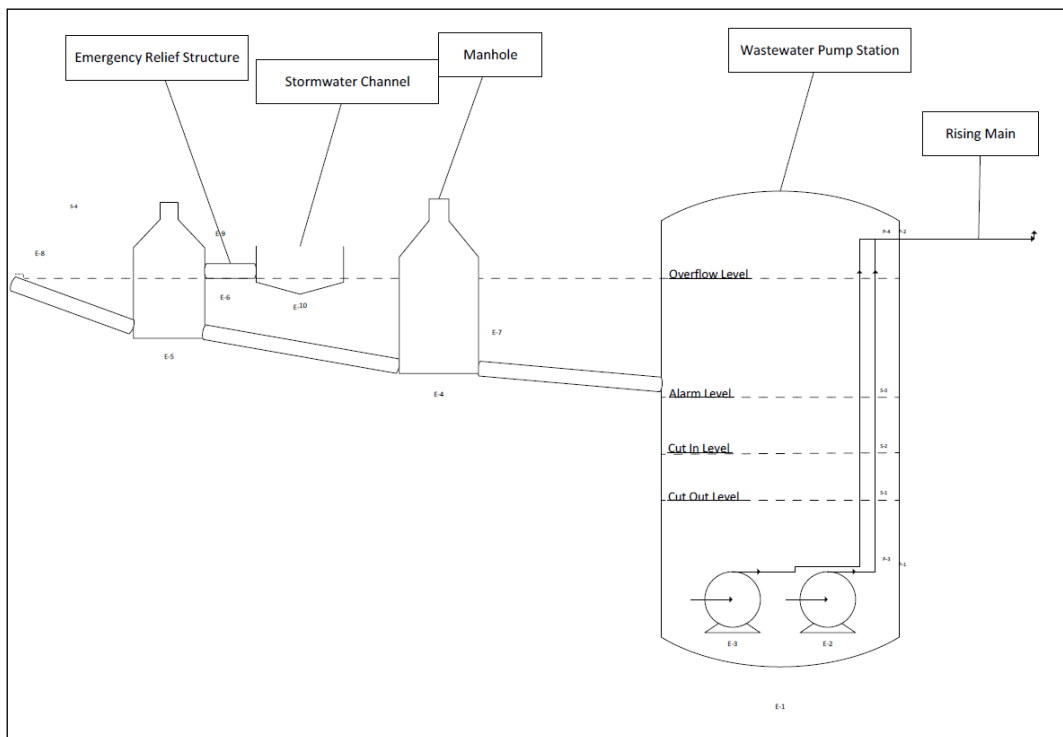


Figure 2.2 Schematic of Wastewater Pump Station Emergency Storage

Under normal dry weather flow conditions the water level would transition between the cut-in and cut-out levels within the station.

In the event of pump station failure the level in the station would continue to rise until an alarm is triggered. Operations staff would then be notified that a problem exists at the station which should allow staff the time to develop a suitable response to manage inflows. This may involve a combination of tankers to manage gravity flows to the station and shutting down upstream stations.

It is important for Water Utility Planning and Operations Divisions to know the exact time available before an overflow will occur to the environment. Dry weather overflows are a license breach reportable to our regulatory authority.



Design guidelines require a minimum of 4 hours emergency storage at all pump stations and 8 hours for pump stations located in sensitive receiving environments. In practice the design guidelines are not always achievable this is why it is important to have accurate information systems to determine available storage and time to overflow.

The pump emergency calculation in MOUSE is invoked by applying an ADP (ADditional Parameter) file to the network simulation containing all the settings for the Pump Emergency Storage Estimation.

Two new sections have been defined for use in the ADP file and named [PUMP\_ESE] and [NODE\_ALARMLEVELS]. The [PUMP\_ESE] includes a sub section named [PUMP\_SECTIONS].

The new sections and parameters are listed in table below.

Table 2.2 ADP file - sections and parameters for configuring up Pump Emergency Storage Estimator

<b>[PUMP_ESE]</b>	
<b>Parameter Setting</b>	<b>Comment</b>
Active = true	true or false
ShortDateFormat = '/', 'DD/MM/YYYY' or ShortDateFormat = '-', 'YYYY-MM-DD'	Optional parameter '<date separator character>', <date format> Default is: '-', 'YYYY-MM-DD'
StopTime = '2014-11-28 14:15:00'	Specification of time when pumps should stop (failure time)
RunMode = 'PumpSections'	'NetWork' or 'PumpSections'
StopUpstreamPumpsDuringFlowRecording = true	true or false - Only used for RunMode = 'PumpSections'
StopUpstreamPumpsDelayMinutes = 15	Delay stopping upstream pumps. [minutes]
OnlyCheckWetWellCriticalLevel = false	true or false - Only used for RunMode = 'PumpSections'
UseAlarmLevels = true	true or false - Controls if 'Alarm Levels' are used
SaveResultFile = true	true or false - Controls writing results to the standard PRF complete results file
PumpESELog = true, 'PumpESELogFile.txt', 0, false	<Switch On/Off>, <LogFileName>, LoggingLevel, <StaticVolumeDetails On/Off> LoggingLevel = 0 (default) or higher for more detailed information StaticVolumeDetails : Controls detailed output of upstream static volume
[PUMP_SECTIONS] Line = '1471746','1475152' Line = '1471746','1474197','1475152' EndSect //PUMP_SECTIONS	Pump Wet Well Node ID's lists
<b>EndSect //PUMP_ESE</b>	

[NODE_ALARMLEVELS]	
Parameter Setting	Comment
UNIT_TYPE = 1	UNIT_TYPE = 1 : SI units, default UNIT_TYPE = 2 : US units (levels in [ft])
NodeAlarmLevelDefault = 80, 1	This setting will be applied for all nodes where the alarm level is not set specifically. <ul style="list-style-type: none"> <li>• Option = 0 : All nodes (default)</li> <li>• Option = 1 : Only pump nodes</li> <li>• NodeAlarmLevelDefault = &lt;%&gt;, Option</li> </ul>
NodeAlarmLevel = '1475634', -1.24 NodeAlarmLevel = '1475152', -0.40 NodeAlarmLevel = '1903657', 1.04 NodeAlarmLevel = '1471746', -0.31 NodeAlarmLevel = '1474197', -2.93	NodeAlarmLevel = '<Node ID>', <Alarm level> where the <Node ID> is the pump wet well node ID The <Alarm level> is in unit [m] when UNIT_TYPE = 1 is set in this section The <Alarm level> is in unit [ft] when UNIT_TYPE = 2 is set in this section
EndSect //NODE_ALARMLEVELS	

When the “//” characters are found the rest of that line is considered a comment and ignored. You may use “//” in front of a parameter to de-activate that specific setting.

The ADP file is created and edited by using a text editor i.e. NotePad. This is the standard procedure for editing the MIKE URBAN ADP file. The ADP file may contain many other sections, which can be set by the user to control various features the MOUSE simulations.

### 2.10.1 [PUMP\_ESE] Section

The parameter “**Active**” activates (“Active = true”) or de-activates (“Active = false”) the PESE analysis. When switched off it is possible to keep the information in ADP file without affecting normal network simulations.

The date format can be optionally controlled by the parameter, “**ShortDateFormat**”. ShortDateFormat = '/', 'DD/MM/YYYY' or ShortDateFormat = '-', 'YYYY-MM-DD'. The default setting used by the MOUSE engine is: '-', 'YYYY-MM-DD', which is used if the parameter is omitted.

The parameter “**StopTime**” is used to set the time when pumping stations failure occurs.

The parameter, “**RunMode**” controls which type of analysis to perform. There are two options available, 'NetWork' (Former: Two Step Mode), or 'PumpSections' (Former: Selected pump mode).

## RunMode = 'Network' – execution sequence

The 'Network' mode has following execution sequences listed below.

This mode does NOT apply the "Pump Sections" lists feature.

- Loads the MIKE URBAN model
- Runs normal simulation until the user specified "Time to stop pumps" is reached
- At this time the following is done to the model for all pumps:
  - The pump capacity is set to zero. (corresponds to that the pump is stopped)
- With these changes to the model the simulation is continued until the specified end time
- During this continued simulation it is monitored if the critical level or alarm level (as specified) is exceeded at the pump wet wells. When the selected level is exceeded it is recorded at which time this happens for each pump. This information is kept available for the next simulation step.
- The MIKE URBAN simulation is terminated
- -----
- The MIKE URBAN model is again loaded and initialized
- Runs normal simulation until the user specified "Time to stop pumps"
- From this time the simulation continues with pumps active
- For all pumps:
  - The pumped volume is recorded during the period from "Time to stop pumps" until the previously recorded time for reaching critical level or alarm level (as specified). This is done individually for each pump.
- The simulation continues until the specified end of simulation time.
- The MIKE URBAN simulation is terminated
- The results are written to the CSV file and to the Log-file

## RunMode = 'PumpSections' – execution sequence

The 'PumpSections' mode has the execution sequence as listed below:

This mode applies the "Pump Sections" lists feature.

- Loads the MIKE URBAN model
- Runs normal simulation until the user specified "Time to stop pumps"
- At this time the following is done to the model for all pump stations mentioned in the current pump section list:
  - The pump capacity is set to zero. (The pump is stopped)
  - The first pump station in the list is "remembered" as the "main" pump station for this analysis

- With these changes to the model setup the simulation is continued until the critical level or alarm level (as specified) is reached. Note: There are two options available for this check, as a user specified setting:
  - The level check is done only at the wet well node for the “main” pump station (the first in the list)
  - The level check is done for all pump stations mentioned in the list
- The simulation will stop once the critical level or alarm level is reached at one of the monitored points
- The time of the level limit exceedance is stored
- The MIKE URBAN simulation is terminated
- - - - - -
- The MIKE URBAN model is again loaded and initialized
- Runs normal simulation until the user specified “Time to stop pumps”
- From this time the simulation continues with the identified “main” pump station active (first pump station in the list)
- For the remaining pump stations in the list there are two options available:
  - All pump stations mentioned in the list are active
  - All pump stations in the list, except the “main” (first) pump station are stopped (The “Stop Upstream Pumps During Flow Recording” option – see below)
- The pumped volume at the “main” pump station is recorded during the period from “Time to stop pumps” until the previously recorded time for reaching critical level or alarm level (as specified).
- The simulation continues until this time is reached.
- The MIKE URBAN simulation is terminated
- The results are written as an added line (row) to the CSV file and to the Log-file

The procedure is repeated for all specified pumping station section lists.

With the parameter **“StopUpstreamPumpsDuringFlowRecording”** it is possible to stop the upstream pumps during recording of the available emergency storage volume when “RunMode” is set to ‘PumpSections’. As an example if an alarm level is reached in a pumping station then the operator can decide to stop upstream pumps as a management action to increase the time before of sewer overflow may occur. However if the upstream pumps are not stopped then the worst situation can be estimated. The settings for the parameter “StopUpstreamPumpsDuringFlowRecording” is either ‘true’ or ‘false’ where ‘true’ is: ‘Stop all pumps in the list except the first listed’ and ‘false’ is: ‘Run all pumps during flow accumulation’.

The parameter **“StopUpstreamPumpsDelayMinutes”** can be used to delay the stopping of the pumps. It is only used for “RunMode” = ‘PumpSections’ and when “StopUpstreamPumpsDuringFlowRecording” = ‘true’. There may be a delay in time from when decision has been taken to stop the pumps to when the pumps are actually stopped in reality. This option can be used to control the delay and is entered in minutes.

“**OnlyCheckWetWellCriticalLevel**” is only used for RunMode = 'PumpSections'. The parameter can have the settings 'true' or 'false' where 'true' is: 'Check CrLev at first pump only' and 'false' is: 'Check CrLev at all pumps in the list'.

“**UseAlarmLevels**” is a new option and controls if alarm levels should be used instead of critical level. The “UseAlarmLevels” can have the settings 'true' or 'false'. If the setting is 'false' then critical level is used instead. The input of alarm levels is provided through separate section in the ADP file.

“**SaveResultFile**” is a new option which controls the writing of results to the standard PRF complete results file. The “SaveResultFile” can have the settings 'true' or 'false'.

“**PumpESELog**” controls the writing to log file. The “PumpESELog” has the settings 'true' and 'false' to switch On/Off, followed by 'LogFileName' and 'LoggingLevel'. LoggingLevel = 0 is default and higher number provide more detailed output. I.e. PumpESELog = true, 'PumpESELogFile.txt', 0.

### [PUMP\_SECTIONS] Sub-section

The [PUMP\_SECTIONS] is a sub-section within the [PUMP\_ESE] section and is used when “RunMode” is set to 'PumpSections'. In this section the pump wet wells node ID are listed.

An example is provided below:

```
[PUMP_SECTIONS]
  Line = '1471746','1475152'
  Line = '1471746','1474197','1475152','1903657'
EndSect //PUMP_SECTIONS
```

### [NODE\_ALARMLEVELS] Section

“**UNIT\_TYPE**” is an option which controls the unit used. The “UNIT\_TYPE” can be set to, UNIT\_TYPE = 1: SI units, default (levels in [m]) or UNIT\_TYPE = 2: US units (levels in [ft]).

“**NodeAlarmLevelDefault**”. This is an optional parameter for easy assigning alarm level to all nodes. The setting will be applied for all nodes where the alarm level is not set specifically.

The syntax is given as **NodeAlarmLevelDefault = <%>, Option**, where

- Option = 0 : All nodes (default)
- Option = 1 : Only pump nodes

and the node alarm level is set as % utilization of:

- Nodes without pump or weir: Distance from bottom level to ground level
- Pump nodes: Distance from highest pump start level to ground level
- Weir nodes: Distance from highest weir crest level to ground level

“**NodeAlarmLevel**”. This is a parameter used for setting the specific alarm level for each pumping station wet well node. The syntax for the definition of the alarm levels is provided below:

**NodeAlarmLevel = '<Node ID>', <Alarm level>**

Where

- The <Node ID> is the pump wet well node ID
- The <Alarm level> is in unit [m] when UNIT\_TYPE = 1 is set in this section
- The <Alarm level> is in unit [ft] when UNIT\_TYPE = 2 is set in this section

An example is given below:

```
NodeAlarmLevelDefault = 80, 1
NodeAlarmLevel = '1475634', -1.24
NodeAlarmLevel = '1475152', -0.40
NodeAlarmLevel = '1903657', 1.04
NodeAlarmLevel = '1471746', -0.31
NodeAlarmLevel = '1474197', -2.93
```

## 2.10.2 Example of Pump Emergency Storage Estimator – ADP file

The ADP file can be configured in a text editor i.e. Notepad. An ADP file has been configured for a sample model and named “PumpESE\_ADP.adp” and listed below for running ‘PumpSections’ mode. The active settings are viewed in bold.

The ‘//’ inactivates a setting as all characters to the right are ignored.

```
[PUMP_ESE]
Active = true // true or false Switch PumpESE feature On/Off
// When switched Off you can still keep the
information // in ADP file while executing normal simulations

// ShortDateFormat = '/', 'DD/MM/YYYY' // Optional parameter.
// ShortDateFormat = '-', 'YYYY-MM-DD' // Optional parameter.
// Can be set as: '-', 'YYYY-MM-DD' or '/', 'DD/MM/YYYY' or
// '/', 'MM/DD/YYYY'
// or similar. '<Date separator char>', '<Date format>'
// The default setting is: '-', 'YYYY-MM-DD' this is used if the
parameter
// is omitted

StopTime = '2011-01-01 06:00:00' // Pump Stop time.
// Format: 'YYYY-MM-DD hh:nn:ss' or date as specified by optional
// 'ShortDateFormat' parameter

StopUpstreamPumpsDuringFlowRecording = true // true or false
// - Only used for RunMode = 'PumpSections'
// True: Stop all pump stations in the list except the first listed
// False: Run all pumps during flow accumulation

StopUpstreamPumpsDelayMinutes = 15
// Delay stopping upstream pump stations. [minutes]
// Only used for RunMode = 'PumpSections'
// Only used when: StopUpstreamPumpsDuringFlowRecording = true

OnlyCheckWetWellCriticalLevel = false // true or false
// - Only used for RunMode = 'PumpSections'
// True: Check CrLev at first pump station only
// False: Check CrLev at all pump stations in the list
```

```

UseAlarmLevels = true           // true or false
//- Controls if 'Alarm Levels' are used

RunMode = 'PumpSections'       // 'NetWork' or 'PumpSections'

SaveResultFile = true         // true or false
// Controls writing results to the standard PRF complete results file

PumpESELog = true, 'PumpESELogFile.txt', 0, false
// PumpESELog = <Switch On/Off>, <LogFileName>, LoggingLevel
// LoggingLevel = 0 (default) or higher for more detailed information
// StaticVolumeDetails : true = Output static volume calculation
// details for upstream links and nodes (Default = False)

[PUMP_SECTIONS]
  //--- Pump Station Wet Well Node ID's lists ---
  Line = '1471746','1475152'
  Line = '1471746','1474197','1475152','1903657'
EndSect //PUMP_SECTIONS
EndSect //PUMP_ESE

[NODE_ALARMLEVELS]
UNIT_TYPE = 1
// UNIT_TYPE = 1 : SI units, default
// UNIT_TYPE = 2 : US units (levels in [ft])

NodeAlarmLevelDefault = 80, 1
// The alarm level can as default be set as % utilization of:
// nodes without pump or weir: distance from bottom level to ground
level
// pump nodes: distance from highest pump start level to ground
level
// weir nodes: distance from highest weir crest level to ground
level
// This setting will be applied for all nodes where the alarm level is
not
// set specifically.
// Option = 0 : All nodes (default)
// Option = 1 : Only pump nodes
// NodeAlarmLevelDefault = <%>, Option

// NodeAlarmLevel = '<Node ID>', <Alarm level>
// where the <Node ID> is the pump station wet well node ID
// The <Alarm level> is in unit [m] when UNIT_TYPE = 1 is set
// The <Alarm level> is in unit [ft] when UNIT_TYPE = 2 is set
NodeAlarmLevel = '1475634', -1.24
NodeAlarmLevel = '1475152', -0.40
NodeAlarmLevel = '1903657', 1.04
NodeAlarmLevel = '1471746', -0.31
NodeAlarmLevel = '1474197', -2.93
EndSect //NODE_ALARMLEVELS

```

