MIKE
Powered by DHI

WEST

User Guide

Tools User Guide

Powering **Water Decisions**

MIKE 2024

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 programs 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 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 in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, 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 jurisdictions 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.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.

CONTENTS

WEST User Guide . . . 11

1 Preface . . . 13
- 1.1 The WEST Tools . . . 13

2 The Graphical User Interface . . . 15
- 2.1 The WEST Environment . . . 15
- 2.2 The Getting Started Sheet . . . 17
  - 2.2.1 Getting Started . . . 17
  - 2.2.2 News Headlines . . . 19
- 2.3 Sheets . . . 19
  - 2.3.1 Layout Sheet . . . 20
  - 2.3.2 Dashboard Sheets . . . 24
- 2.4 System Windows . . . 26
  - 2.4.1 The Block Library . . . 26
  - 2.4.2 The Control Center . . . 27
  - 2.4.3 The Block Summary . . . 27
  - 2.4.4 The Model Explorer . . . 28
  - 2.4.5 The Block Details . . . 31
  - 2.4.6 The Properties Window . . . 33
  - 2.4.7 The Logging Window . . . 33
  - 2.4.8 The Layers Window . . . 34
  - 2.4.9 The Overview Window . . . 35
- 2.5 The Application Menu . . . 36
  - 2.5.1 WEST Options . . . 37
- 2.6 The Menu Bar . . . 40
  - 2.6.1 Home | Editing Group . . . 41
  - 2.6.2 Home | Clipboard Group . . . 41
  - 2.6.3 Home | Help Group . . . 41
  - 2.6.4 Insert | Sheets Group . . . 42
  - 2.6.5 Insert | Connections Group . . . 42
  - 2.6.6 Insert | Illustrations Group . . . 42
  - 2.6.7 Insert | Input Group . . . 43
  - 2.6.8 Insert | Output Group . . . 43
  - 2.6.9 Project | Properties Group . . . 44
  - 2.6.10 Project | Miscellaneous Group . . . 44
  - 2.6.11 Project | Advanced Group . . . 45
  - 2.6.12 Project | Virtual Experiments Group . . . 45
  - 2.6.13 View | Windows Group . . . 46
  - 2.6.14 View | General Group . . . 46

2.6.15 View | Show/Hide Group . . . 46
2.6.16 View | Sheets Group . . . 47
2.6.17 View | Animation Group . . . 47
2.6.18 View | Refresh Group . . . 47
2.6.19 Tools | Support Group . . . 47
2.6.20 Tools | Applications Group . . . 48
2.6.21 Format | Arrange Group . . . 48
2.6.22 Format | Rotate Group . . . 48
2.6.23 Format | Flip Group . . . 49
2.6.24 Format | Align Group . . . 49
2.7 The Status Bar . . . 50
2.8 The Context Menu . . . 50
2.9 The Interface Tools . . . 53
2.9.1 Types of Data . . . 54
2.9.2 The Influent Characterisation . . . 55
2.9.3 Scalar Data Input . . . 63
2.9.4 Vector Data Input . . . 64
2.9.5 Merging Data Sources . . . 66
2.9.6 The Effluent Characterisation . . . 68
2.9.7 Scalar Data Output . . . 71
2.9.8 Vector Data Output . . . 71
2.10 The Properties of the GUI Objects . . . 72
2.10.1 Properties of the Layout Sheet . . . 72
2.10.2 Properties of a Block . . . 72
2.10.3 Properties of a Rectangle . . . 76
2.10.4 Properties of a Textbox . . . 76
2.10.5 Properties of a Picture . . . 77
2.10.6 Properties of a Line . . . 78
2.10.7 Properties of a Connection . . . 78
2.10.8 Properties of a Dashboard Sheet . . . 79
2.10.9 Properties of a Slider Pane . . . 79
2.10.10 Properties of an Input (Output) Field Pane . . . 81
2.10.11 Properties of a Radiobutton Pane . . . 83
2.10.12 Properties of a Combobox Pane . . . 85
2.10.13 Properties of a Checkbox Pane . . . 87
2.10.14 Properties of a Plot Window . . . 89
2.10.15 Properties of a Table Pane . . . 94
2.11 Shortcuts . . . 96

3 The WEST Model . . . 99
3.1 The Input Model . . . 99
3.2 The Plant Model . . . 100
3.2.1 Flattened Model . . . 100
3.2.2 Parameters . . . 100
3.2.3 Top-Level Parameters . . . 101
3.2.4 Top-level Interface Variables . . . 102
3.2.5 Calculator Variables . . . 104
3.3 The Output Model . . . 105

4 Experiment Types . . . 107
4.1 Steady-State and Dynamic Experiment . . . 108
4.1.1 Simulation Properties . . . 108
4.1.2 Analysis Properties . . . 116
4.2 Local Sensitivity Analysis Experiment . . . 123
4.2.1 Analysis Properties . . . 123
4.3 Global Sensitivity Analysis Experiment . . . 127
4.3.1 Analysis Properties . . . 127
4.4 Parameter Estimation Experiment . . . 133
4.4.1 Analysis Properties . . . 134
4.5 Scenario Analysis Experiment . . . 138
4.5.1 Analysis Properties . . . 138
4.6 Uncertainty Analysis Experiment . . . 145
4.6.1 Analysis Properties . . . 145
5 Use of WEST . . . 153
5.1 How to Manage Projects . . . 153
5.1.1 Create a New Project . . . 153
5.1.2 Open an Existing Project . . . 155
5.1.3 Open a Sample . . . 155
5.1.4 Close a Project . . . 156
5.2 How to Create a Plant Layout . . . 157
5.2.1 Place Blocks on the Layout Sheet . . . 157
5.2.2 Assign a Block to a Layer . . . 158
5.2.3 Rename a Block . . . 158
5.2.4 Resize a Block . . . 159
5.2.5 Rotate a Block . . . 160
5.2.6 Flip a Block . . . 161
5.2.7 Replace the Icon of a Block . . . 161
5.2.8 Replacing a Block (and underlying model) . . . 162
5.2.9 Connect two Blocks . . . 163
5.2.10 Align Blocks . . . 166
5.2.11 Change the Model Selection of a Block . . . 167
5.2.12 Define Top-Level Parameters . . . 168
5.2.13 Define Scalar Top-Level Interface Variables . . . 169
5.2.14 Define Calculator Variables . . . 171
5.2.15 How To Set Up a Controller . . . 172
5.3 How to Manage Dashboard Sheets . . . 172
5.3.1 Add or Delete a Sheet . . . 173
5.3.2 Add or Delete an Input or Output Object . . . 173
5.3.3 Arrange Input and Output Objects Within a Sheet . . . 173
5.3.4 Populate Input and Output Objects Within a Sheet . . . 175
5.4 How to Manage Input Data . . . 176
5.4.1 Wastewater Input / Standard ASM vector . . . 176
5.4.2 Wastewater Input / Custom vector . . . 178
5.4.3 Data Input . . . 179
5.4.4 Vector Input . . . 180

5.4.5 Create a Fractionation Model . . . 181
5.5 How to Manage Simulation Inputs . . . 185
5.5.1 Use of a Slider . . . 185
5.5.2 Use of an Input Field . . . 186
5.5.3 Use of a Radiobutton . . . 186
5.5.4 Use of a Combobox . . . 187
5.5.5 Use of a Checkbox . . . 187
5.5.6 Remove a Slider, Field, Radiobutton, Combobox or Checkbox . . . 187
5.6 How to Manage Output Data . . . 188
5.6.1 Wastewater Output / Standard ASM vector . . . 188
5.6.2 Wastewater Output / Custom vector . . . 188
5.6.3 Data Output . . . 189
5.6.4 Vector Output . . . 189
5.6.5 Create a De-fractionation Model . . . 190
5.7 How to Manage Simulation Outputs . . . 193
5.7.1 Use of an Output Field . . . 193
5.7.2 Use of a Table . . . 193
5.7.3 Use of a Plot . . . 194
5.7.4 Use of a Gauge . . . 201
5.7.5 Remove Output Field or Data Series From a Table, a Plot or a Gauge . . . 203
5.7.6 Output Data to File . . . 203
5.7.7 Export Results to Excel . . . 205
5.8 How to Run Dynamic Simulations . . . 206
5.8.1 Initialise the Model(s) . . . 206
5.8.2 Execute the Experiment . . . 207
5.9 How to Create a Report . . . 207

6 Technical Specifications . . . 211
6.1 Prerequisites . . . 211
6.2 How to use the VC++ compiler with WEST 32-bit . . . 211
6.3 How to use WEST 2020 64-bit with Visual Studio Community . . . 211

WEST Tools User Guide . . . 213

7 Block Editor . . . 215
7.1 Description . . . 215
7.2 Quick User Guide . . . 217

8 Data Editor . . . 219
8.1 Description . . . 219
8.2 Quick User Guide . . . 219

9 Designer . . . 221
9.1 Description . . . 221
9.2 Quick User Guide . . . 221

10 Model Editor . . . 223
10.1 The WEST model Library . . . 225
10.2 The User Interface . . . 225
10.3 Examples . . . 227
10.3.1 How to create a new Block Library . . . 227
10.3.2 How to create a new Instance . . . 228
10.3.3 How to create a new Category . . . 228
10.3.4 How to import an existing Category . . . 229
10.3.5 How to generate code . . . 231

11 Unit Editor . . . 233
11.1 Description . . . 233
11.2 Quick User Guide . . . 233

Index . . . 235

WEST USER GUIDE

1 Preface

This document illustrates the use of WEST and comprises the following chapters:

Chapter 2 presents the Graphical User Interface (GUI) of WEST. The properties of graphical objects, input/output widgets, experiments, integrator, etc. are dealt with in a homogenous way in the WEST Environment; i.e. through the Properties pane. A special tool, called influent/effluent tool is available in WEST, which assists the user in the generation of an input file and of input and output models.

Chapter 3 presents the WEST Plant Model (the counterpart of the graphical layout of the plant), which is comprised of three main sub-models: the Input, the Plant and the Output sub-models. The possibility of defining parameters and interface variables at the level of the overall Plant model (top-level parameters and variables) is also illustrated.

Chapter 4 presents the different types of experiments available in WEST, starting from the base experiment type, i.e. Steady-State and Dynamic Simulation (which is called Objective Evaluation Experiment), to the advanced virtual experiment types that can be built on top of the base experiment.

Chapter 5 presents the practical use and illustrates the various features of the software.

1.1 The WEST Tools

WEST consists of the following separate applications:

- **WEST**:
  The main application where one can graphically set up a plant layout and execute dynamic simulations and advanced virtual experiments, such as scenario analysis or parameter estimation.

- **Model Editor**:
  Modelling environment that offers a syntax-aware Code Editor (for direct editing of models in the msl language) and a Matrix Editor (for editing the biochemical models through and ad-hoc graphical interface and automatic code generation).

- **Block Editor**:
  Application that allows for creating and managing icon and palette libraries for use in WEST.

- **Unit Editor**:
  Small application that allows for visualising and managing the WEST unit table (used to perform unit conversions in WEST).

- **Data Editor**:
  Application that allows for managing data, e.g. input files.

- **Designer**:
  Application that allows for designing a treatment plant according to one of the available templates and based on conventional design rules (i.e. ATV or Metcalf & Eddy).

This document covers the first application.

The WEST Tools User Guide briefly illustrates the remaining applications.

2 The Graphical User Interface

The “new” generation of WEST (starting from WEST2011) uses a completely new approach to the User Interface, as compared to older versions (WEST 3.x) where the steps to be undertaken to set up and run a dynamic simulation would take place in separate ‘environments’. WEST consists now of a single interface both for layout building and for running simulations.

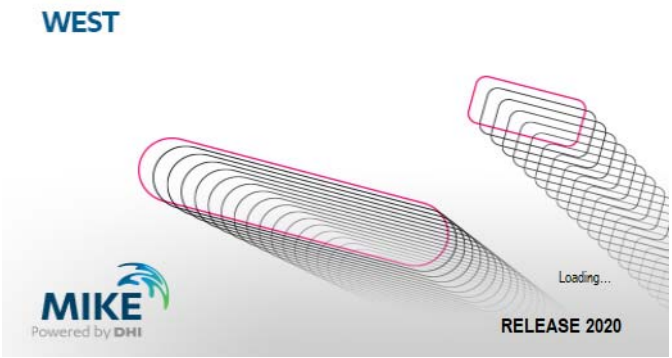


Figure 2.1 Splash screen of WEST

2.1 The WEST Environment

The WEST User Interface consists of the following elements (Figure 2.2):

- **Getting Started Sheet** (Figure 2.3):
  The Getting Started sheet is displayed at start-up and every time a Project is closed; it exposes functionalities to create or open a Project; to retrieve miscellaneous information related to Help and News; and to edit the project’s Description property. It can be accessed from within a Project from the View | System menu.

- **Sheets**:
  Four types of Sheets are available in WEST, i.e. the Layout, the Interface, the Dashboard and the Web-Browser Sheets:
  - The *Layout Sheet* is the area where the Blocks are to be dropped from the Block Library to set up the model layout.
  - The *Interface Sheets* expose a dedicated environment for the generation and subsequent manipulation of input files and Fractionation/Defractionation models (cf. Section 2.9)
  - The *Dashboard Sheets* act as containers for Input and Output widgets, i.e. Sliders, Fields, Radio buttons, Combo boxes, Check boxes, Plots, Tables and Gauges.

  - A *Web-Browser Sheet* is started when one of the external tools available under the Home | Help menu, or under the News Headlines window is accessed.

- **System Windows**:
  Provide a view on the properties of the object selected on a Sheet (the Properties, the Block Details and the Block Summary panes), a way to navigate through the model structure of the experiment (the Model Explorer), a way to inspect the activity of WEST (the Logging pane), a view on the Layout Sheet (the Layers and Overview pane), a graphical representation of the Model Library (the Block Library), and a means to run the experiments (the Control Center pane).

- **Application Menu**:
  Gives access to the basic operations (such as New Project, Open, Save, etc.) and to the WEST Options.

- **The Menu bar**:
  Offers a quick way to access all the functionalities of WEST in the form of menu items (i.e. Home, Insert, Project, View, Tools and Format).

- **The Status bar**:
  Displays short messages (on the left-hand side), the current experiment type and mode and the Zoom tool (on the right-hand side).

Sheets and System Windows can be repositioned in the main window and can be made invisible (closed): all but the Layout Sheet can be *docked* in various positions or left floating, by dragging their caption bar.

Invisible Sheets or System Windows can be activated (i.e. made visible) through the respective menu items in the View | Windows menu.

Remark: Floating windows may go lost when opening a project using a display with lower resolution (than the one used to save the project). They can however be re-arranged in a cascade by clicking on the View | Cascade button.

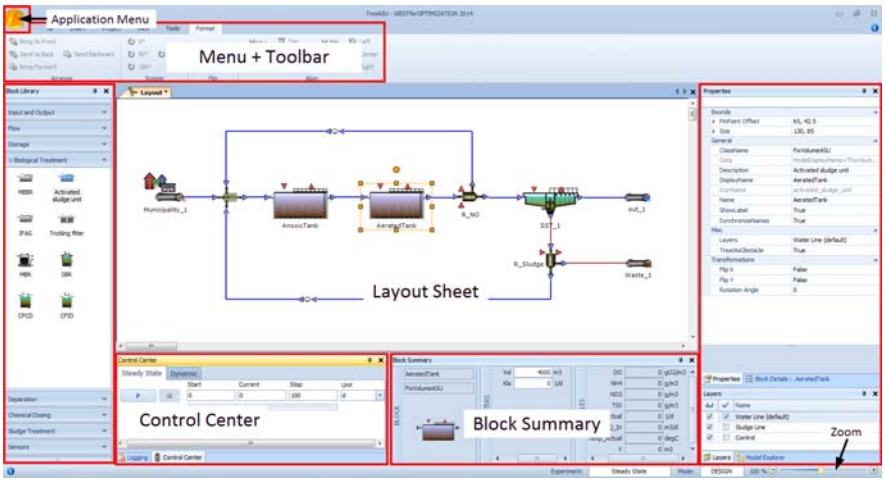


Figure 2.2 The default WEST Environment

2.2 The Getting Started Sheet

The Getting Started Sheet of WEST (Figure 2.3) comprises the Project Overview and the News Headlines panes.

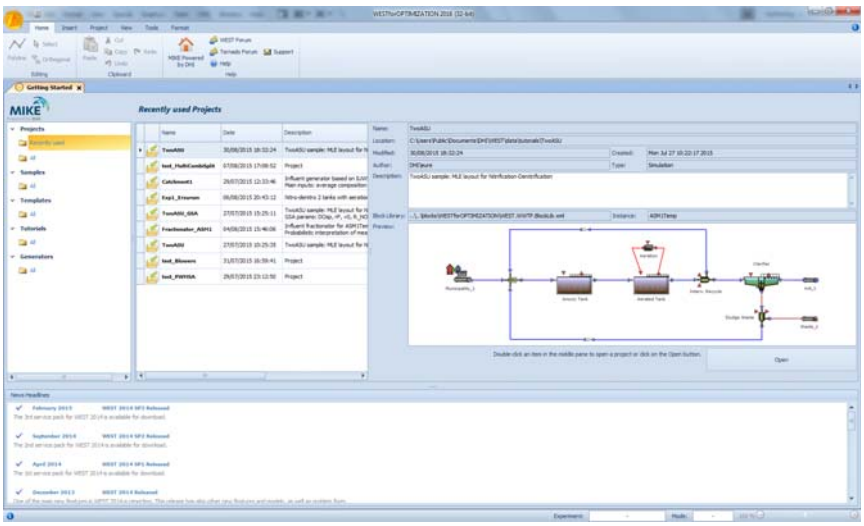


Figure 2.3 The Start-up view of WEST

2.2.1 Getting Started

The Getting Started Sheet (Figure 2.4) consists of three frames which display a tree view of Project categories (left), a list of the Projects available under

the selected category (centre) and an overview of the properties of the selected Project (right).

The default Project *categories* are:

- **Projects**:
  Contains two *folders* that give access to the Recent Projects or to All Projects, respectively.
- **Samples**:
  Contains a number of sample cases.
- **Templates**:
  Templates (including the basic Blank Project) can be used as the starting point to construct complex plant layouts. Templates include: Bardenpho with 4 and 5 stages; Ludzack-Ettinger; modified Johannesburg; etc.
- **Tutorials**:
  These are fully worked out test cases that illustrate the use of WEST and the different experiment types. Each tutorial is illustrated in detail in the Tutorial's guide.
- **Generators**:
  Projects that can be used to generate a synthetic influent file.

Once a *category* is selected in the tree view, the corresponding collection of Projects is displayed in the frame in the centre. Two alternative views are possible:

- the Card View (default): the appearance can be modified using the small toolbar atop
- the Grid View: possibly more convenient when one deals with a large collection of projects.

Switching from one view to the other can be done through the WEST Options (cf. 2.5.1 'WEST Options').

Projects can be sorted on Name and Time.

As one browses through the collection of Projects, a set of summary properties of the current Project is displayed in the frame on the right: these include the Name of the Project, a Description, the Block Library and the specific Model Instance used to create the Project and a snapshot of the Layout, amongst others.

The project's Description can be edited without opening the project explicitly.

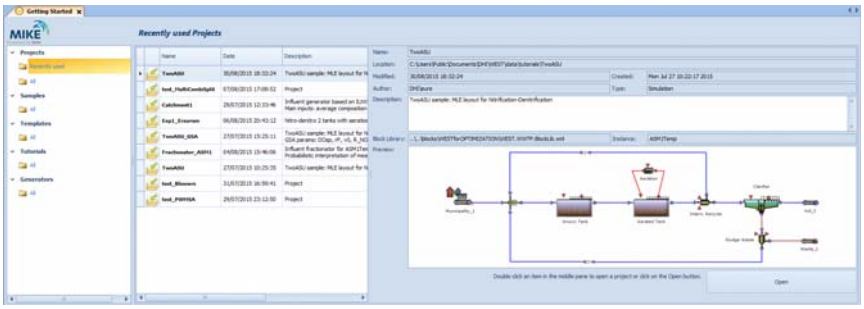


Figure 2.4 The Getting Started pane

2.2.2 News Headlines

The News Headlines present a number of links to the News items of the MIKE Powered by DHI website (http://www.mikepoweredbydhi.com) that are automatically updated from the DHI server.

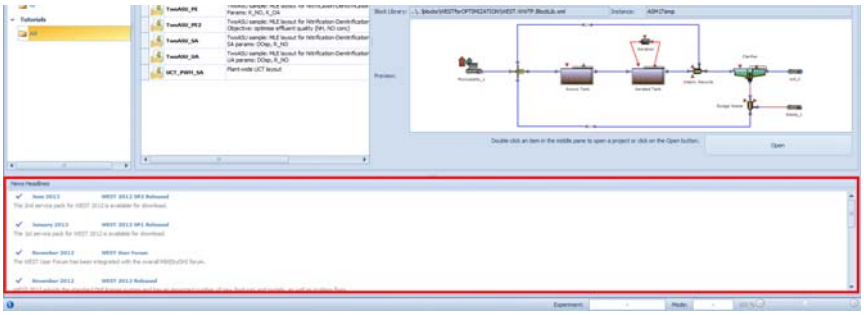


Figure 2.5 The News Headlines frame

2.3 Sheets

Four classes of Sheets are available in WEST:

- **Layout Sheet**:
  Contains the Plant Layout.
- **Interface Sheets**:
  Contain a set of tab pages to create the input file and, if relevant, the influent fractionation model (Influent Sheet) and the effluent defractionation model (Effluent Sheet). Every Input (Municipality, Industrial, Truckload, Data or Vector Input) and Output Block (Effluent, Waste, Data or Vector Output) has a corresponding Sheet.
- **Dashboard Sheets**:
  Contain Input and Output widgets.

- **Web Browser Sheets**:
  Can display: the MIKE Powered by DHI website, the WEST and the Tornado Forum and the software documentation.

A Project may only contain one Layout Sheet but can have several Sheets of the other classes.

2.3.1 Layout Sheet

The Layout Sheet is the drawing area that opens up as a new Project is started; Blocks are to be dropped from the Block Library and connections between them established in order to set up a Plant Layout (and the corresponding Model in the background, Section 3.2).

The Block

A Block is the graphical representation of the model of a unit process. It consists of an **Icon,** a certain number of **Terminals** and a **Label** (Figure 2.6).

The Terminals are anchor points that are used to graphically draw a connection between two Blocks.

Hovering over a Terminal displays a pop-up window showing the following properties of the terminal (Figure 2.8): Name, Type, Description, Causality and Maximum Number of Connections.
The Description (to be entered as Terminal Property in the Block Editor) allows for better clarifying the meaning of the terminal in case its name would not be sufficiently descriptive.

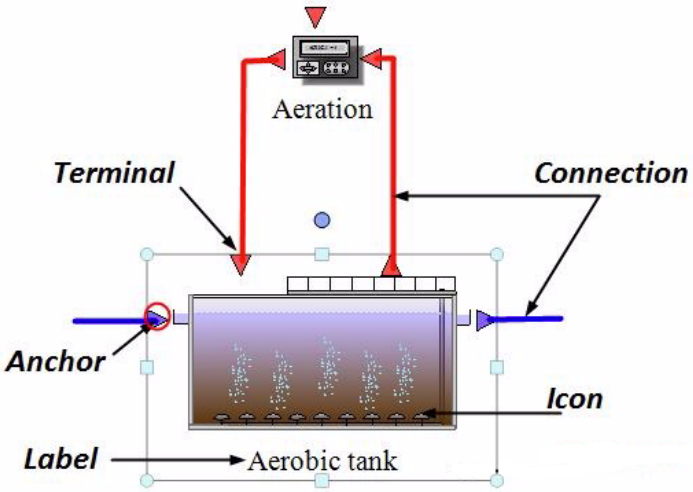


Figure 2.6 Elements of a Block and its relation to other Blocks

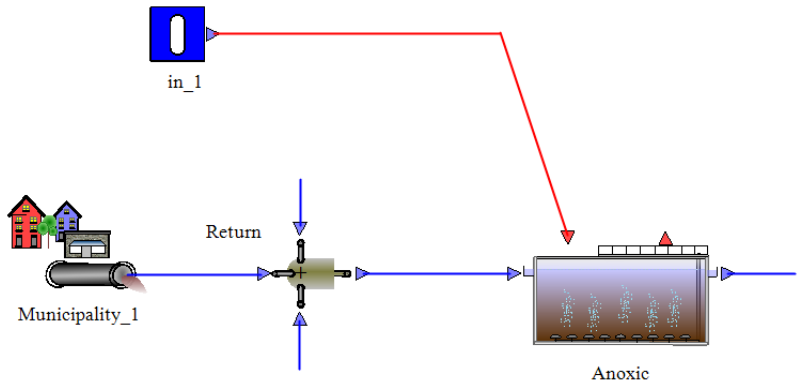


Figure 2.7 Connecting an Input Block to another Block

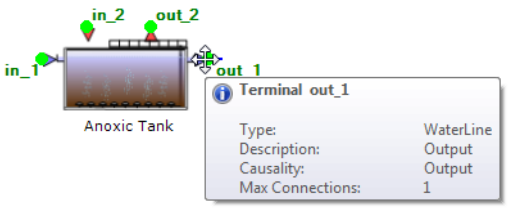


Figure 2.8 Pop-up window showing the terminal's properties

Interface Blocks

Interface Blocks represent the input to and output of the plant (model), respectively.

Different interface blocks are available in WEST (Figure 2.9).

An Interface of type “Component Vector” denotes the influent (effluent) of the plant and requires that the characteristics of the influent (effluent) be specified.

Further distinction is made within the Component Vector category of inputs, between a **municipal** and an **industrial** type input (influent) and between a continuous and a batch input (Figure 2.9).

A scalar Interface denotes the input (output) of a Manipulated Variable (e.g. a sludge waste schedule, the temperature, etc.).

A vector Interface can be used in the scope of hierarchical modelling (i.e. re-using layouts as blocks in other layouts). It creates a top-level vector Interface Variable that uses the components of one of the Categories of the active Instance.
Note that the VectorInput wizard also allows for Data Import (pure data, no fractionation nor C/F conversion). This feature can be used for unit testing the

sub-layout, but is not relevant when re-using the sub-layout in the scope of hierarchical modeling

Remark: Up until the current version (2020), the different wastewater input and output blocks (i.e. Municipal, Industrial and Truckload on the one hand, Effluent and Waste on the other) may be used to visually connote different streams but have the same exact behaviour, i.e. open the Influent and Effluent Tool Sheets, respectively.

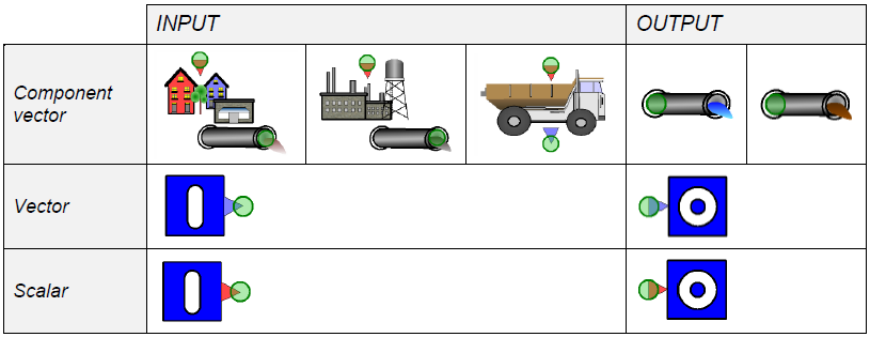


Figure 2.9 Types of Input and Output Blocks

Ordinary Blocks

A generic (i.e. non-interface) Block in WEST has one or more associated models which reside in the Model Library - and which can be selected through its ClassName property in the Properties pane.

Loop Breaker Block

A special type of Block, the Loop Breaker, is needed to break what is termed an **algebraic loop**.

Algebraic loops are the result of calculations in which the input is a function of the output of the model, thus creating a loop. Equation (2.1) shows an example of an algebraic loop:

$$\begin{aligned} a &= b + c \\ b &= d - e \\ c &= 2a + 1 \end{aligned} \tag{2.1}$$

Algebraic loops are most often introduced by a graphical loop in the layout (typically, a recycle). However, data loops do not result in an algebraic loop because the link between the measured value and the control handle is generally indirect.

A (numerical) integrator is not capable of solving this type of equations, consequently a specific solver would have to be used - which however would

slow down the simulation speed considerably. A more efficient solution is to break the algebraic loop manually, by introducing a Loop Breaker Block onto the graphical loops, as shown in Figure 2.10.

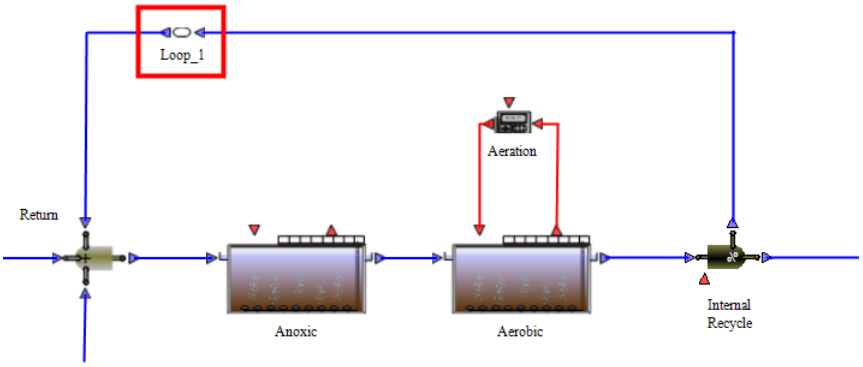


Figure 2.10 Position of a Loop Breaker on a recycle flow

Loop Breakers can automatically and transparently be inserted into a layout. Three properties (EnableControlConnections, InsertLoopBreakers, Loop-Breaker) control the behaviour, all of which are accessible through the layout properties. It is possible to complement manual Loop Breaker placement with automatic insertion. All to be used with care.

Terminals

Terminals are the *ports* of a Block. They represent input and output nodes for the connections between Blocks and are graphically represented by incoming and outgoing arrows in the Block Icon.

There are five classes of Terminals in WEST, i.e. Generic, Water, Sludge, Gas and Data.

Connections

A Connection represents the flow of physical or data quantities between two Blocks.

Connections are drawn between the *outgoing* Terminal (i.e. a Terminal with *causality* 'Output') of one Block to the *incoming* Terminal (i.e. a Terminal with *causality* 'Input') of another Block. Evidently, a connection cannot be set between two incoming Terminals.

Furthermore, only Terminals of the same type can be connected: the resulting Connection will be of the same type, i.e. Generic, Water, Sludge, Gas and Data, and will have the colour specified in the WEST Options, i.e. black, blue, brown, green and red, respectively.

2.3.2 Dashboard Sheets

Dashboard Sheets enable the user to interact with the Experiment, i.e. by manually altering values of Parameters and/or Manipulated Variables and by visualising the results of the Simulation.

Dashboard Sheets act as *containers* for Input and Output widgets; however, the same Sheet may contain **both types of widgets**:

- **Input widgets** are Sliders, Fields, Radio buttons, Combo- and Check boxes
- **Output widgets** are Plots, Tables, Fields and Gauges

Other types of I/O objects are available which cannot be used in combination with Sheets. An overview is shown in the following Table 2.1.

Table 2.1 I/O objects available in WEST

I/O Channel	INPUT		OUTPUT	
	Graphical	Data	Graphical	Data
Slider	X			
Field	X		X	
Radiobutton	X			
Combobox	X			
Checkbox	X			
Plot			X	
Table			X	
Gauge			X	
Animation			X	
File		X		X
Buffer				X

Input widgets

These can be associated with either a Manipulated Variable or a Parameter and are therefore used to manually alter a value **during a simulation** (in the case of a Manipulated Variable) or **between two simulations** (in the case of a Parameter).

- Slider
  horizontal bar with a thumb that can move within a given range
- Field
  allows for typing in a single numerical value

- Radiobutton
  allows for exposing a limited number of alternatives, associated to (mutually exclusive) values of the underlying model quantity
- Combobox
  same as a set of radiobuttons, a combobox exposes a limited number of alternatives, but in the form of a drop-down menu. The alternatives associated to (mutually exclusive) values of the underlying model quantity
- Checkbox
  sets a value of '0' or '1' (unchecked or checked state) to an underlying model quantity

Output widgets

These are associated with one or more Model Variables and are therefore used to display the output of a simulation while the simulation is running.

- Table
  grid of rows (time points) and columns (time series). The first column is automatically assigned to the independent variable "time"
- Plot
  a variety of plot types is available in combination with each type of dataset, as shown in Table 2.2 and Figure 2.11.
- Field
  allows for displaying the current value of the selected variable, in the form of a numerical value
- Gauge
  allows for showing the current value of the selected variable, in the form of a moving needle

Table 2.2 Types of plot for each dataset type

Type of plot	Time series X(t)	Current value	Y(X)	Z(X,Y)	Histogram
Line	X	X	X		
Fast line	X	X	X		
Point	X	X	X	X	
Column	X	X	X	X	
Bar		X			
Pie		X			
Area		X			
Surface		X		X	

Table 2.2 Types of plot for each dataset type

Type of plot	Time series X(t)	Current value	Y(X)	Z(X,Y)	Histogram
Color Grid		X		X	
Vertical					X
Horizontal					X

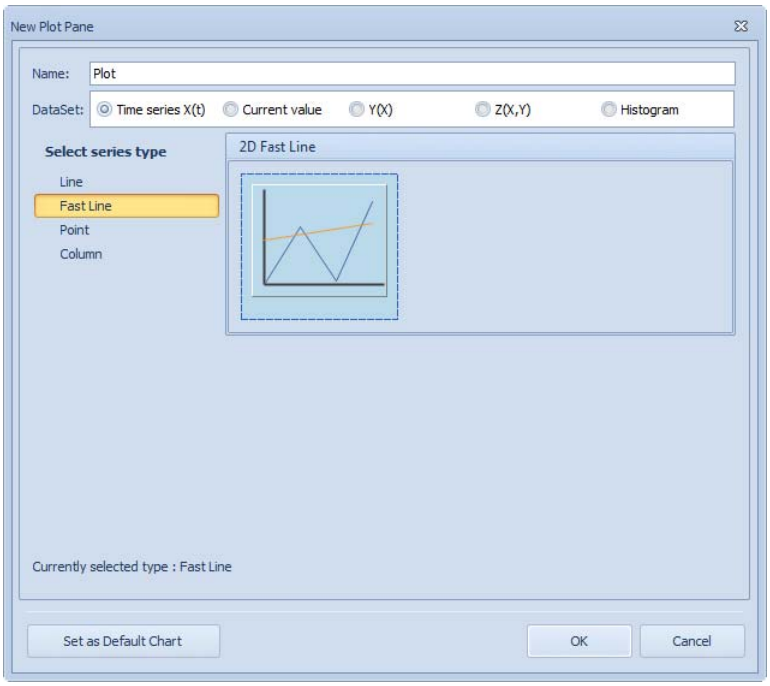


Figure 2.11 Creating a new Plot

2.4 System Windows

Windows of this category provide views on some properties of the WEST Model Library or ways to interact with the simulation engine.

All System Windows are accessible through the View | Window | System menu or via their respective short-cut (Table 2.4 in section 2.11 Shortcuts *(p. 96)*).

2.4.1 The Block Library

The Block Library displays palettes containing the Blocks available to create the Model Layout.

A reference to a particular Palette Library is present in the BlockLib.xml file that is to be selected when creating a new Project (see Section 5.1.1). One can also set the default Block Library that is to be used for all new Projects, through the WEST Options item of the Application Menu.

2.4.2 The Control Center

The Control Center is the means (Figure 2.12) to control the simulations: a simulation (or a virtual experiment) can be started, paused and stopped and its progression is dynamically shown in the progress bar and as Current time (or Run No.).

The F6 and F7 keys can be used as shortcuts for Start / Pause and Stop respectively.

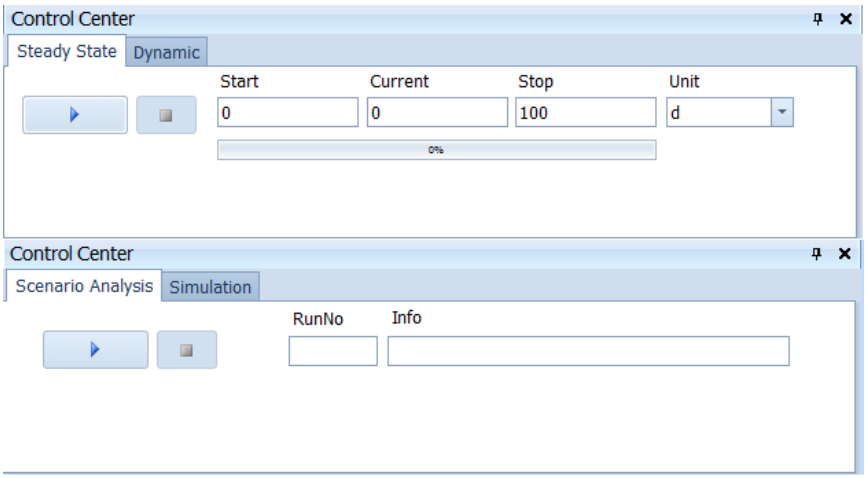


Figure 2.12 The Control Center window for the base experiment (top) and for an advanced virtual experiment (bottom)

The simulation time horizon, i.e. start and stop time, and the time unit are to be set here.

2.4.3 The Block Summary

The Block Summary presents information on the Block that is selected on the Layout Sheet (Figure 2.13); it is subdivided into three sections, i.e. 'Block', 'Parameters' and 'Variables'. The 'Block' area shows a miniature of the Block, the name of the Block and the model and that has been selected; the 'Parameters' and 'Variables' display a selection of model parameters and variables respectively that are considered to be most relevant for the specific Block.

The Parameters and Variables sections belong to the “Info” tab page.

If a parameter or a derived state variable is flagged as favourite in the MSL code of the model, its value (current value for a parameter and current or initial value for a derived state variable) can be changed through this pane rather than through the Block Details pane.

The list of favourite parameters and variables can be modified:

- at the project level, by dragging and dropping the parameters and variables of interest from the Block Details pane. By doing so, the view will change automatically from “Standard” to “Custom”.
- at the level of the model, by assigning the annotation ‘is_favorite=1’ to the parameter or variable of interest - using the Model Editor tool

In “Custom” mode, project-dependent favourite parameters and variables can be removed (through the local menu associated with the blue arrow in the top left corner).

The display unit of a particular item can also be selected in the Block Summary pane. Synchronization of units between Block Details, Block Summary and Control Center is ensured.

The tab page “Alerts” displays the alerts generated by the Animation channel during the simulation:

- Time: reports the time at which the violation happened
- Quantity: shows the variable that exceeded the given threshold
- Violation: specifies the type of violation

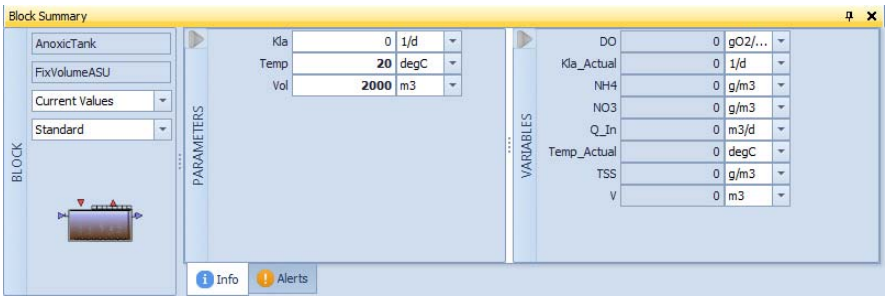


Figure 2.13 The Block Summary

2.4.4 The Model Explorer

The Model Explorer offers a graphical representation of the hierarchical model structure (illustrated in Chapter 3) in the form of a tree view (Figure 2.14). It consists of a topmost level which represents the Project itself and from which all process nodes stem, each corresponding to a sub-model of the overall model.

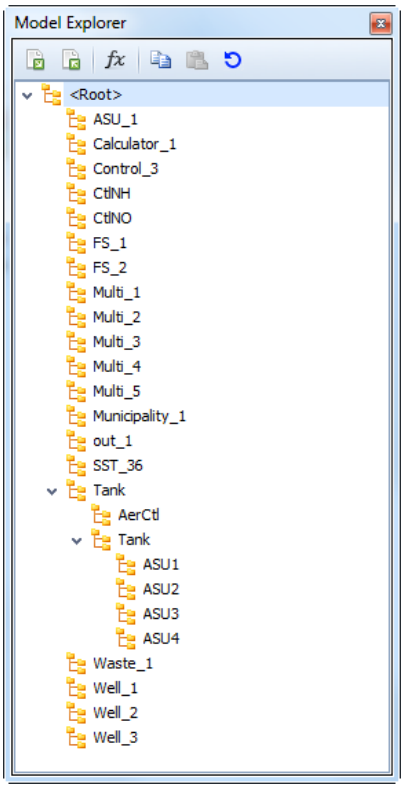


Figure 2.14 The Model Explorer window

The following buttons are available in the toolbar of the Model Explorer pane:

- **Store** allows saving the Initial- or Current values of the Parameters, Input Variables and Derived Variables into a new Set within a specified Parameter library (cf. Figure 2.15).
- **Load** allows retrieving a Set of parameters from a specified Parameter library.
- **Rescale** allows rescaling the initial values (of parameters, input variables or derived state variables) of certain quantities by providing a factor and an offset. The quantities are to be specified using a regex pattern (see http://www.ecma-international.org/publications/standards/Ecma-262.htm).
- **Copy Values** takes a snapshot of the state (parameters, input variables and derived state variables) of a node
- **Paste Values** transfers values previously stored (copied) into a node
- **Reset Values** resets the state (parameters, input variables and derived state variables) of a node to its default

All buttons work in a hierarchical fashion, i.e. on all models below the selected node. So if the current node is one of the sub-models, it stores,

loads, rescales or resets the values of this model only; if the current node is the “Root”, it does it for all the blocks of the layout.

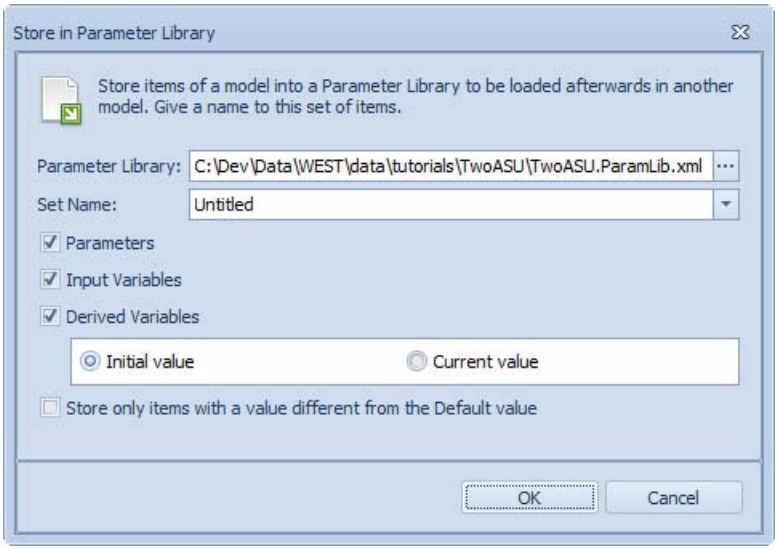


Figure 2.15 Save a new Set of model parameters into a ParamLib.xml file

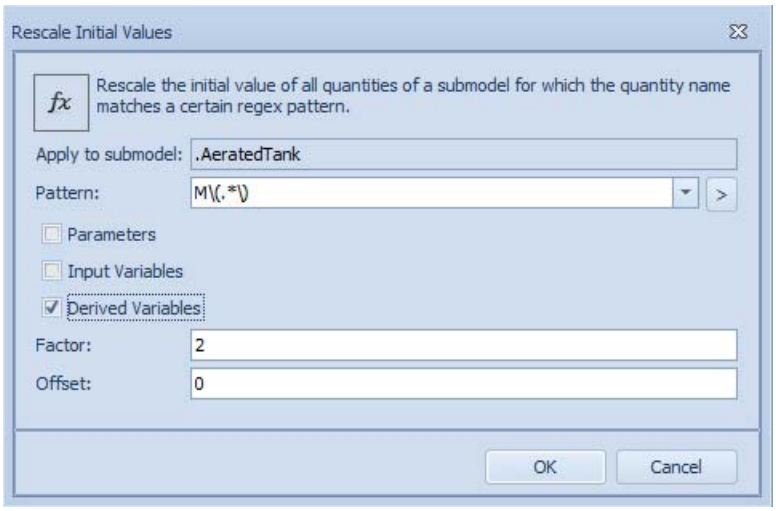


Figure 2.16 Rescale Initial Values dialogue: in the example shown here, all derived state variables of the sub-model “AeratedTank” named “M(*)” (i.e. the mass vector) are going to be multiplied by a factor 2

2.4.5 The Block Details

The Block Details window gives access to the Parameters and Variables of the Model underlying the current Block that is selected either in the Layout Sheet or in the Model Explorer.

The Description tab page shows the description associated with the model in the MSL code (read-only field). The Parameters and Variables tab pages can be viewed as such, i.e. separately (default) or next to each other within a single tab page, by toggling the minus button in the top-right corner. In case multiple Blocks are selected, the Block Details pane shows the common parameters and variables (default view); or all parameters and variables of every Block (by toggling the Show All/Common button in the top-right corner).

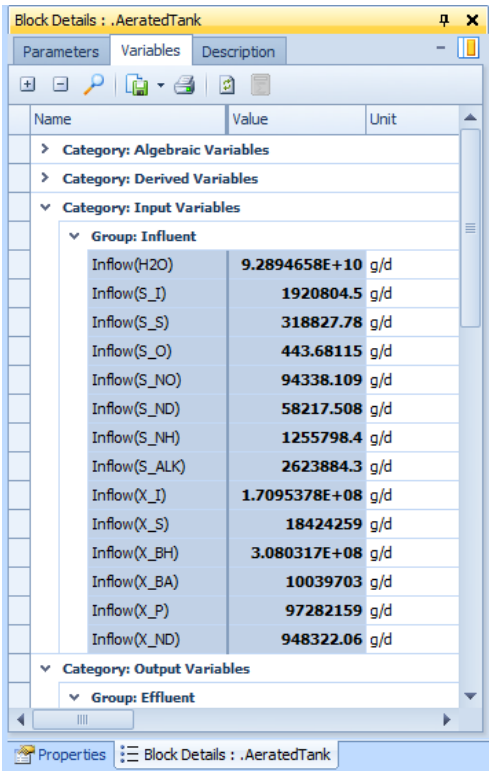


Figure 2.17 The Block Details window

For each parameter or variable the following properties are available:

- **Name**:
  the name of the parameter or variable.
- **Value**:
  the current value as applied in the experiment.

- **Initial value**:
  (only for derived state variables) the value used at the start of the simulation.
- **Default value**:
  the default value that is assigned to a parameter or variable in the model library.
- **Unit**:
  The unit used to display the value.
- **Lower bound**:
  The minimum value that is allowed for the parameter or variable.
- **Upper bound**:
  The maximum value that is allowed for the parameter or variable.
- **Description**:
  (not exposed by default) the description of the parameter or variable.
- **Category**:
  The type of parameter (parameter or manipulated variable) or variable (input, output, algebraic variable, derived state variable and independent variable).
- **Group**:
  A property for grouping parameters and variables.
- **Fixed**:
  (not exposed by default) a property that indicates whether the parameter or variable is fixed (i.e. not editable)
- **Hidden**:
  (not exposed by default) indicates whether a parameter or variable is visible or not in the GUI. Invisible quantities can be shown by checking the “Show hidden quantities” option in the application menu / Options
- **Block**:
  (not exposed by default) shows the name of the block that a parameter or variable belongs to
- **Derivative**:
  (not exposed by default) shows the derivative of a derived state variable

Properties that are not visible can be retrieved by right-clicking on any column header; selecting the Column Chooser item and dragging-dropping the property into the Block Details pane.

The toolbar of the Parameters and Variables tab page contains the following buttons:

to collapse or expand rows

to expose the description of each quantity, immediately below its identifier

to export the content of the current tab page to Excel, HTML or text file

to print the content of the current tab page

(Parameters tab page only) to “promote” the selected parameters to top-level (Section 3.2.3): only enabled if one or more parameters are selected

(Variables tab page only) to update the current values

(Variables tab page only) to open the Calculator Variables dialogue (Section 3.2.5): only enabled at the Root level

Standard column operations (e.g. sorting, grouping, filtering, etc.) can be accesses by right-clicking on any column header

2.4.6 The Properties Window

This is a very general window that serves the purpose of giving access to the Properties of the object that is currently selected on one of the Sheets, such as a Block or an Input or Output item or one of their components (e.g. a data series).

The Properties window of each object of the WEST GUI is examined in detail in Section 2.10.

2.4.7 The Logging Window

The Logging window (Figure 2.18) provides a convenient interface to inspect the messages issued by Tornado (the generic back-end at the heart of WEST) or by the GUI itself.

The Logging window consists of a grid view with three columns:

- **Category**:
  denotes the 'type' of message, i.e. Error, Warning or Information. Each type is represented by an icon.
- **TimeStamp**:
  indicates the date and time at which the log event has been issued.
- **Message**:
  displays an extensive message associated to the log event, e.g. the type of error that has occurred.

Additional columns can be enabled through the “Column Chooser” of the standard column context menu (right-click on any column header):

- **Help**: shows a direct link to the documentation where further explanation can be found for the specific log event
- **File**: file in which the event (error or warning) has occurred
- **Line**: line number that fired the event (error or warning)

The toolbar of this pane contains the following buttons:

to clear the logbook

to export the content of the logbook to Excel, HTML or text file

to print the content of the logbook

to enable a scroll lock

to expose all messages or filter only essential ones

Standard column operations (e.g. sorting, grouping, filtering, etc.) can be accesses by right-clicking on any column header.

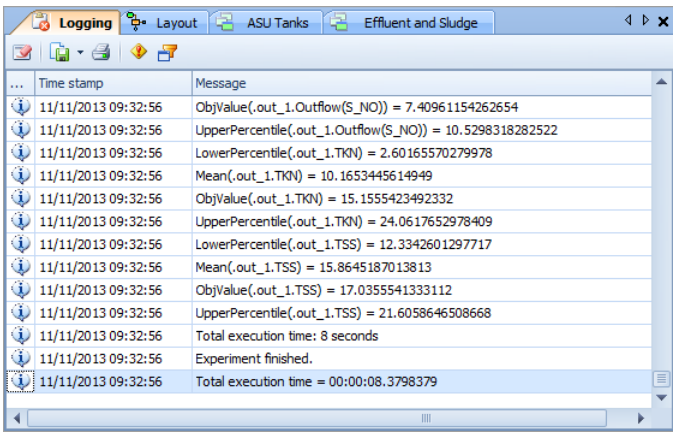


Figure 2.18 The Logging window

2.4.8 The Layers Window

An important feature of WEST is the possibility of using layers to set up the plant layout. The default layers are Water Line, Control and Sludge Line.

The Water Line is the Layer activated, i.e. visible and enabled (Figure 2.19) by default. This means that all Blocks that are placed on the Layout will be

assigned by default to the Water Line, unless another layer is activated, i.e. made the current layer.

Layers may then be independently visualised by (de)selecting the respective Visibility option in the Layers window. Layers can also be merged, by right-clicking on a layer and choosing the Merge option.

Blocks (and their connections) can be re-assigned to a different layer through the Layers property in the Properties pane.

Irrespectively of the Layer that a Block is associated to, the overall model will evidently treat all units as part of the same general layout.

The layer that a Block is assigned to becomes a *property* of the Block – and can be changed at any time, through the Properties pane.

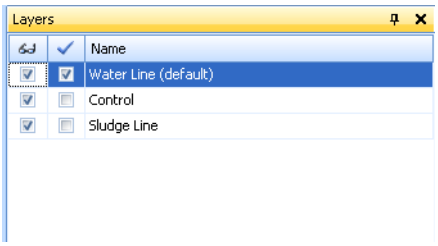


Figure 2.19 The Layers window

2.4.9 The Overview Window

The Overview window shows a miniature of the working area and allows moving the current view, i.e. the sub-area that is to be visualised on the Lay-out Sheet.

It provides two functionalities, i.e. **zooming** and **panning** as shown in Figure 2.20.

The red frame defines the area that is currently visible on the Layout Sheet. By selecting one of the corner handles of the red frame, a second re-sizable frame is displayed that can be expanded or reduced, resulting in zooming out or in respectively (left in the Figure).

By selecting the frame itself (that is: any point inside the frame), the frame can be moved, resulting in another area of the total 'working area' being visualised in the Layout Sheet (right in the Figure).

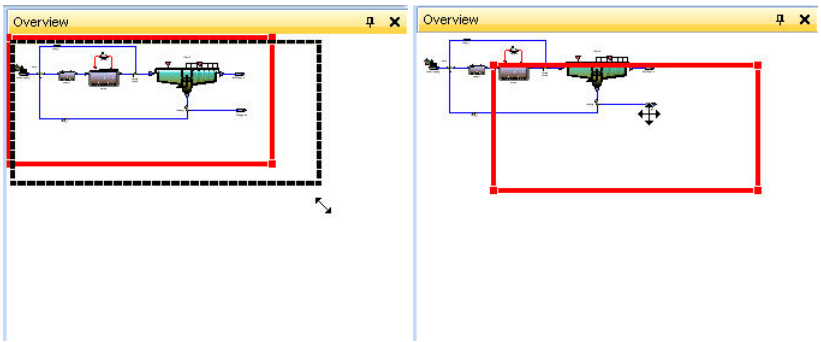


Figure 2.20 Zooming and focusing the layout through the Overview window

2.5 The Application Menu

The Application menu (Figure 2.21) gives access to the following functionalities:

- **New**:
  start a new Project.
- **Open**:
  open an existing Project.
- **Save**:
  save the current Project.
- **Save As**:
  save the current Project under another name (and location).
- **Close**:
  close the current Project.
- **WEST Options**:
  access some advanced options (Section 2.5.1).
- **Exit WEST**:
  quit the application.
- **Recent Projects**:
  start one of the recently used projects.

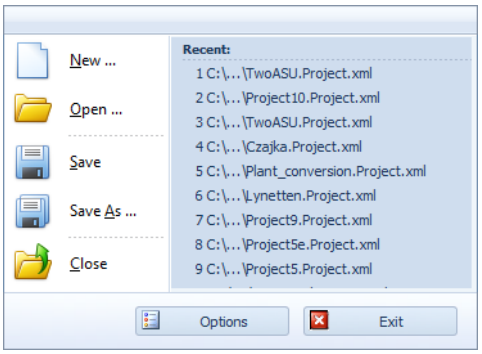


Figure 2.21 The Application menu

2.5.1 WEST Options

The WEST options are grouped in the following categories:

- **Popular**:
  access to a set of WEST options that may need to be changed, more frequently than others.
- **Layout**:
  access to some graphical options for the Layout Sheet.
- **Advanced**:
  access to the advanced settings.

Popular

The ‘Popular’ panel (Figure 2.22) enables you to:

- select the language - should more than one dictionary be enabled with your licence
- limit the number of lines displayed in the logbook
- specify the format used to display numbers and test it
- hide/unhide quantities (parameters and variables labelled as “hidden” in the model code) in the Block Details pane
- set the specific product of the WEST product suite that is to be used at start-up and to enable/disable the selection dialogue at start-up
- automatically copy the end state of a Steady-State simulation to the initial state of a Dynamic Simulation
- choose how to visualise the list of projects in the Getting Started pane, i.e. Card - vs. Grid View
- choose the number of recent projects to be shown in the Getting Started

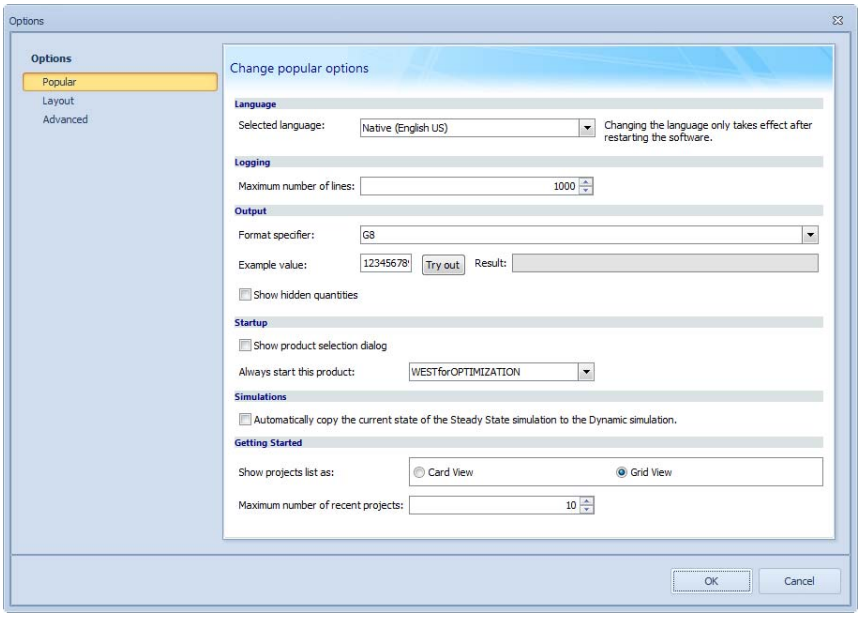


Figure 2.22 The Popular WEST options

Layout

The ‘Layout’ panel (Figure 2.23) presents the overview of the colour codes used for connections (Section 5.2.9).

The default settings are as follows:

- **Generic**:
  black, e.g. for the direct connection between an Input and Output Unit.
- **Control**:
  red, e.g. connections from/to terminals of type Control.
- **Water Line**:
  blue, e.g. all connections from/to terminals of type Water Line.
- **Sludge Line**:
  brown, e.g. all connections from/to terminals of type Sludge Line.
- **Gas Line**:
  green, e.g. all connections from/to terminals of type Gas Line.

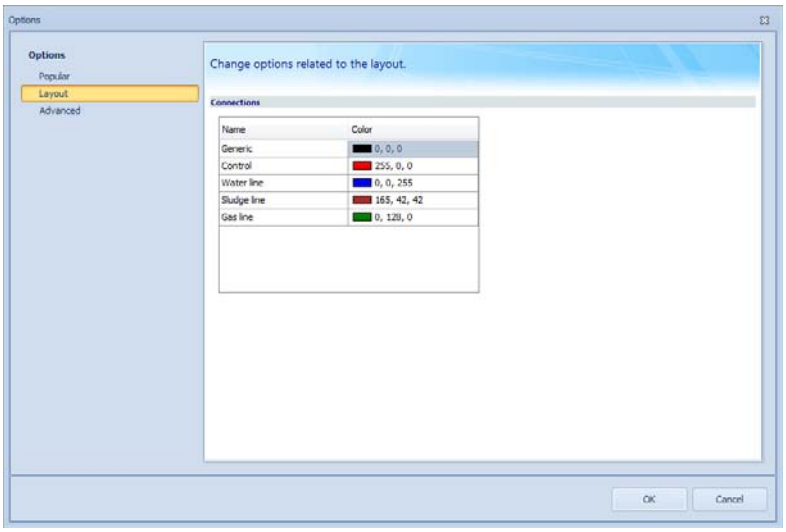


Figure 2.23 The Layout properties of the WEST Options

Advanced

WEST 'Advanced' settings (Figure 2.24) are grouped in the following categories:

- **Data Path**:
  indicates the location where all user data (i.e. Projects, Block libraries, Samples, Templates) are stored and retrieved from.

- **File locations**:
  it is crucial that the location of the following three folders be specified:
  - Projects: folder where new Projects are to be saved.
  - Templates: folder where the Templates reside.
  - Samples: folder where the Samples reside.
  - Tutorials: folder where the Tutorials reside.

- **Default Block Libraries**:
  independent default Block Libraries can be configured for the different products of the WEST product suite.

- **Visualization Interval**:
  visualization interval for the update of Dashboard widgets.

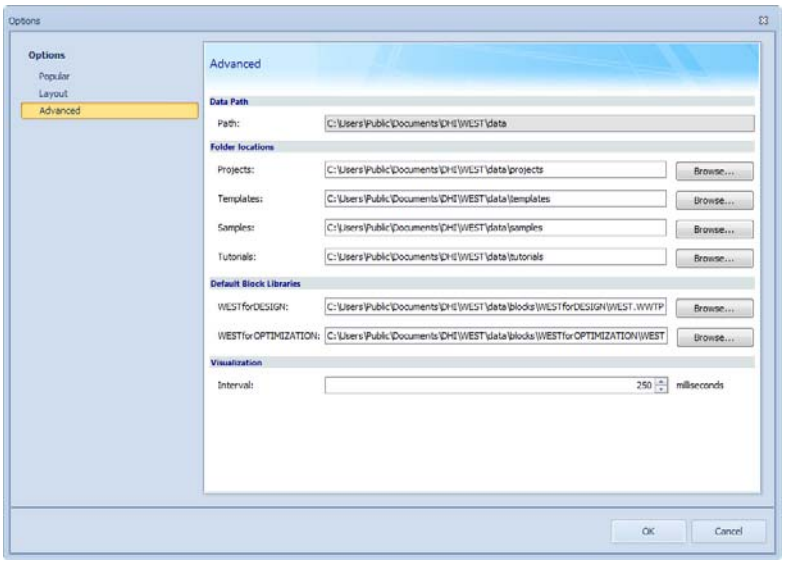


Figure 2.24 The Advanced properties of the WEST Options

2.6 The Menu Bar

The following menu bars (ribbon bars) are shown in the WEST GUI:

- **Home**:
  contains basic editing and help tools.
- **Insert**:
  it exposes the tools to connect Blocks, to add Dashboard Sheets, etc.
- **Project**:
  gives access to a number of Project Properties, i.e.: the Project Description, the Simulation Properties (integrator and integrator settings; and output settings for files, fields, plots and tables) and the Analysis Properties, as well as to the advanced experiments.
- **View**:
  gives access to the settings of the Layout sheet, to the available System Windows and Sheets and to the overall *look-and-feel* of the GUI.
- **Tools**:
  retrieves additional information regarding the configuration of WEST and Tornado, and contains the menu to launch the four external tools: the Block-, the Data-, the Model- and the Unit Editor and the Designer.
- **Format**:
  contains all the functionalities related to the manipulation of Block Icons on the Layout Sheet, e.g. rotation, alignment, etc. and of the objects on the Dashboard Sheets.

2.6.1 Home | Editing Group

This group exposes the following functionalities:

- **Polyline**:
  to draw a line connecting two Blocks on the Layout Sheet; it accepts anchor points; and the angles formed by the segments of the polyline are not bound to 90 degrees.
- **Select**:
  to select an object on the Layout Sheet - it corresponds to the standard mouse pointer. Also necessary to draw a selection area around multiple blocks, for multi-selection.
- **Orthogonal**:
  to draw a line connecting two Blocks on the Layout Sheet; it creates anchor points that are necessary to ensure that the angles formed by the segments of the polyline be 90 degrees.

2.6.2 Home | Clipboard Group

The Clipboard group gives access to the usual functionalities of copy-paste and select:

- **Paste**:
  (Ctrl+P) paste the selection to the Layout Sheet.
- **Cut**:
  (Ctrl+X) cut an object (or group thereof) from the Layout Sheet.
- **Copy**:
  (Ctrl+C) copy an object (or group of objects) from the Layout Sheet.
- **Undo**:
  (Ctrl+Z) to undo the last action
- **Redo**:
  (Ctrl+Y) to redo the last action
- **Select All**:
  (Ctrl+A) to select all the objects in the Layout Sheet

2.6.3 Home | Help Group

The Help group exposes shortcuts to the following external resources:

- **MIKE Powered by DHI Website**:
  to open a browser Sheet on the MIKE Powered by DHI page.
- **WEST Forum**:
  to open a browser Sheet in the MIKE by DHI Forum – the WEST Forum can be found in the category Cities.

- **Tornado Forum**:
  to open a browser Sheet in the MIKE by DHI Tornado Forum (password protected).
- **Help**:
  to open a Sheet containing links to the various documentation items, release notes and support contact details
- **Support**:
  to send an email to the Technical Support team

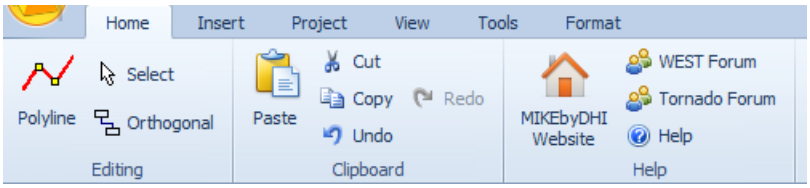


Figure 2.25 The Home menu

2.6.4 Insert | Sheets Group

Dashboard Sheet can be added and removed by clicking on respective buttons.

2.6.5 Insert | Connections Group

Lines connecting Blocks on the Layout Sheet can be drawn in the form of polygonal or orthogonal lines by clicking on respective buttons.

2.6.6 Insert | Illustrations Group

The following types of illustration items can be added to the Layout Sheet:

- **Textbox**: editable text box
- **Rectangle**: rectangular frame
- **Picture**: picture box that can contain an image
- **Pencil**: free drawing

An example of the use of these graphical elements is shown in Figure 2.26.

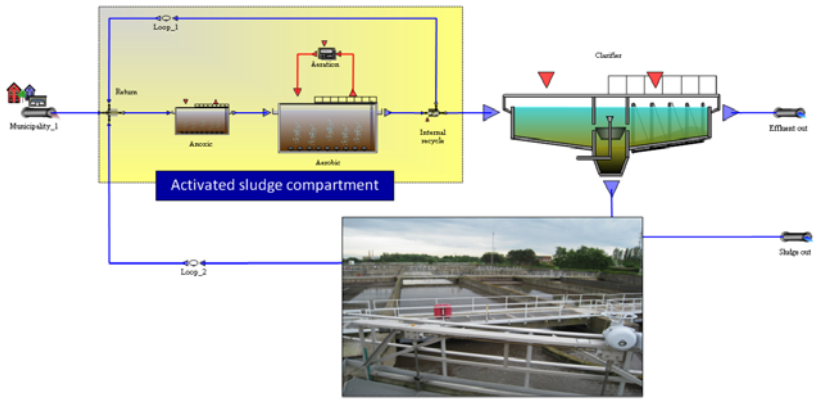


Figure 2.26 The TwoASU sample with additional graphical elements

2.6.7 Insert | Input Group

The Input group exposes the five types of *Input* widgets that can be added to Dashboard Sheets and that are used to manually control Model Parameters and/or Manipulated Variables:

- **Slider**: cursor control (track bar)
- **Field**: editable input field
- **Radiobutton**: set of options in the form of radio-buttons
- **Combobox**: set of options in the form of drop-down lists
- **Checkbox**: set of options in the form of check-boxes

2.6.8 Insert | Output Group

The Output group exposes the four types of *Output* widgets that can be added to Input/Output Sheets and that are used to visualise the output of the simulation:

- **Plot**: graph
- **Table**: table
- **Field**: output field
- **Gauge**: displays the current value of the variable(s) by means of a needle or pointer that moves along a calibrated scale

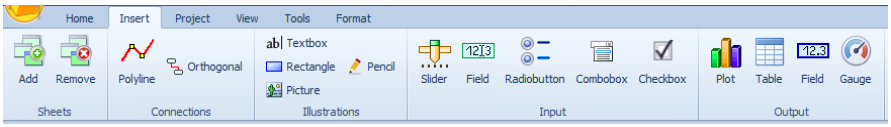


Figure 2.27 The Insert menu

2.6.9 Project | Properties Group

The Properties group gives access to the following:

- **General**:
  allows for editing the Project Description
- **Simulation**:
  allows for
  - selecting the integrator and its settings (Solver tab page, Figure 4.1)
  - choosing whether bounds should be checked and the simulation should stop when bounds are violated (default settings are “yes” and “no” respectively)
  - selecting the settings of the different outputs, i.e. File, Buffer, Field, Plot, Table, Gauge and Animation (Output tab page, Figure 4.1)
  - artificially slowing down the simulation, to mimic real-time mode
  - setting up the various Animation types, i.e. Line Thickness and Colour, Labels and Alerts;
- **Analysis**:
  allows for the definition of the analysis properties, i.e. quantities that are to be computed for the given experiment. This is key for the advanced virtual experiments - see e.g. Section 4.5.1 (Analysis Properties).

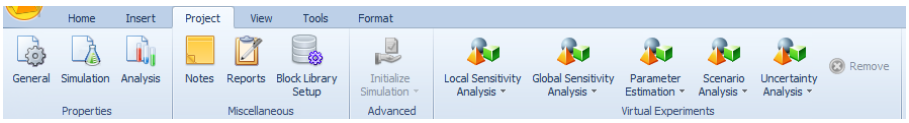


Figure 2.28 The Project menu

2.6.10 Project | Miscellaneous Group

The following project tools are available:

- **Notes**:
  pops up the Notes window, which can be used as an advanced text editor to store all kinds of notes pertaining to the project, including images and tables; all standard editing features are available and the sheet can be saved in various formats, e.g. Text, Richtext, Word, etc.

- **Reports**:
  starts up the Manage Reports dialogue (cf. Section 5.9) to create custom reports.
- **Block Library Setup**:
  to inspect or change the Project settings pertaining to the Block Library, i.e. the Block Library itself, the Instance and the Instance Settings (cf. Figure 2.29).

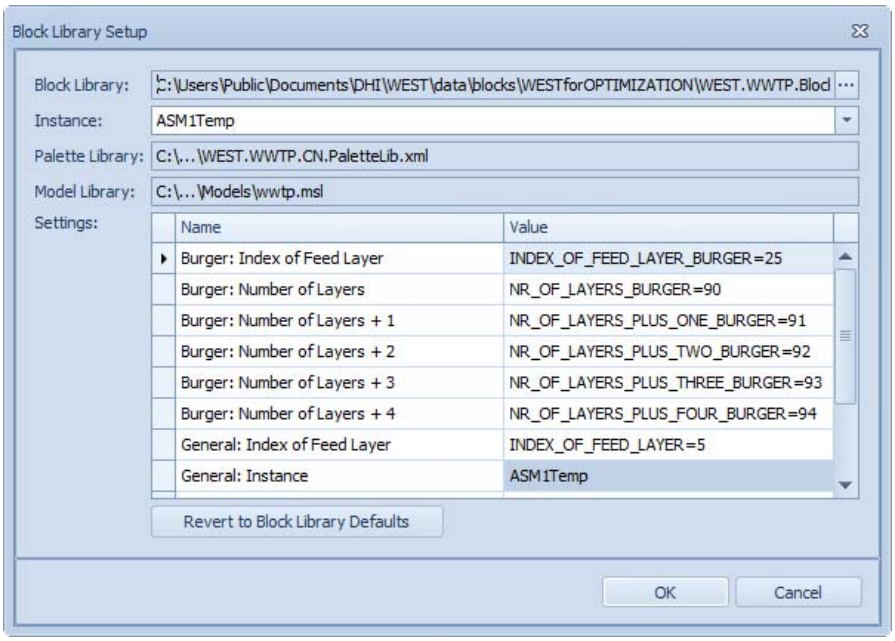


Figure 2.29 The Block Library setup dialogue

2.6.11 Project | Advanced Group

If the checkbox ‘Automatically copy the current state …’ in the Popular tab page of the WEST Options (Figure 2.22) is not checked, four options will be available under the “Initialize Simulation” button:

- From Steady-State to Dynamic
- From Dynamic to Steady-State
- From Steady-State to Steady-State
- From Dynamic to Dynamic

2.6.12 Project | Virtual Experiments Group

The following advanced virtual experiment types are available:

- Local Sensitivity Analysis (LSA)
- Global Sensitivity Analysis (GSA)
- Parameter Estimation (PE)
- Scenario Analysis (SA)
- Uncertainty Analysis (UA)

Only one *advanced* virtual experiment can be built on top of the base experiment - being either the Steady-State or the Dynamic Simulation. Therefore once an *advanced* virtual experiment is created on top of a Simulation, all other virtual experiment buttons are disabled.

Should a different *advanced* virtual experiment be needed, the current one needs to be removed by clicking on the “Remove” button.

Refer to Chapter 4 ‘Experiment Types’ for a detailed illustration of the Properties pertaining to each experiment type.

2.6.13 View | Windows Group

Hidden System Windows and Sheets can be activated through their respective buttons and sub-items in the Windows group.

2.6.14 View | General Group

The following functionalities are available:

- **Look-and-feel**:
  to select a specific look-and-feel for the application
- **Full screen**:
  allows for conveniently working with large layouts - without the hassle of manually clicking away all windows and sheets except for the Layout sheet
- **Cascade**:
  to be used to retrieve floating windows that may have gone lost when opening a project using a display with a lower resolution

2.6.15 View | Show/Hide Group

The Show/Hide group exposes the following functionalities:

- **Ruler**:
  (un)hides the rulers on the Layout Sheet.
- **Grid lines**:
  (un)hides the gridlines on the Layout Sheet.

- **Snap to Grid**:
  (de)activates the snap-to-grid option for the Blocks on the Layout Sheet.

2.6.16 View | Sheets Group

It contains the following buttons:

- **Design mode**:
  switches the view of an Dashboard Sheet to design mode whereby the objects contained in the Sheet can be repositioned
- **Organize**:
  to reorganise the content of the Dashboard Sheets by moving objects from one to the another
- **Clear All Series Data**:
  to clear all the output widgets (Plots, Tables, ..)

2.6.17 View | Animation Group

Allows for toggling between the plain layout (animation layer hidden) and the layout with animation output.

2.6.18 View | Refresh Group

Allows for refreshing the content of dashboard widgets that are associated with data files.

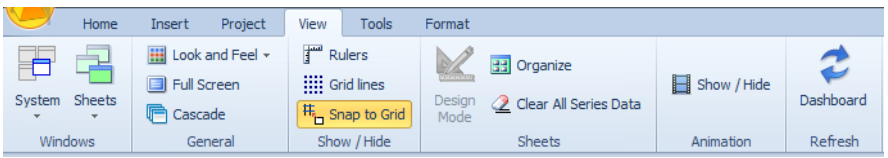


Figure 2.30 The View menu

2.6.19 Tools | Support Group

The Support group exposes the following tools:

- **Tornado Shell**:
  activates the command line toolbox.
- **Event Viewer**:
  monitors events recorded in the Application, Security, and System logs.

- **Diagnostics**:
  executes the WEST Diagnostics and displays the output; this can be copied in the body of an e-mail and sent to the MIKE by DHI Technical Support team (mike@dhigroup.com).

2.6.20 Tools | Applications Group

The Applications group gives access to the other WEST tools:

- Block Editor
- Data Editor
- Designer
- Model Editor
- Unit Editor

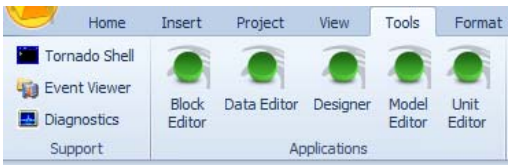


Figure 2.31 The Tools menu

2.6.21 Format | Arrange Group

The Arrange group contains the following items:

- **Bring to Front**:
  brings the selected object to the front on the Layout Sheet.
- **Send to Back**:
  sends the selected object to the back on the Layout Sheet.
- **Bring Forward**:
  brings the selected object forward on the Layout Sheet.
- **Send Backward**:
  sends the selected object backward on the Layout Sheet.

2.6.22 Format | Rotate Group

The Rotate group contains the following items:

- **0°**:
  rotates the selected object by 0° with reference to its initial position.
- **90°**:
  rotates the selected object by 90° with reference to its initial position.

- **180°**:
  rotates the selected object by 180° with reference to its initial position.
- **270°**:
  rotates the selected object by 270° with reference to its initial position.

2.6.23 Format | Flip Group

The Flip group contains the following items:

- **Horizontal**:
  flips the selected object horizontally on the Layout Sheet.
- **Vertical**:
  flips the selected object vertically on the Layout Sheet.

2.6.24 Format | Align Group

The Align group contains the following items:

- **Horizontal Terminals**:
  aligns two or more Blocks horizontally with reference to their terminals, i.e. along a horizontal connection.
- **Vertical Terminals**:
  aligns two or more Blocks vertically with reference to their terminals, i.e. along a vertical connection.
- **Top**:
  aligns two or more Blocks to the top with reference to their Icon (incl. the Label).
- **Middle**:
  aligns two or more Blocks to the middle (horizontal median axis) with reference to their Icon (incl. the Label).
- **Bottom**:
  aligns two or more Blocks to the bottom with reference to their Icon (incl. the Label).
- **Left**:
  aligns two or more Blocks to the left with reference to their Icon (incl. the Label).
- **Center**:
  aligns two or more Blocks to the centre (vertical median axis) with reference to their Icon (incl. the Label).
- **Right**:
  aligns two or more Blocks to the right with reference to their Icon (incl. the Label).
- **Straight line**:
  automatically inserts anchor points on a straight connection

- **Align to Grid**:
  a layout selection can be automatically aligned to the grid - which is important in order to obtain decent line thickness animation results (non-aligned terminals & connections will lead to unwanted staircase effects).

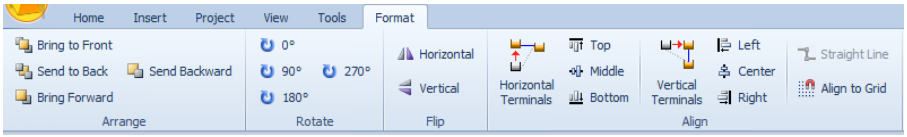


Figure 2.32 The Format menu

2.7 The Status Bar

The Status bar consists of the following elements:

- **Information**:
  (left) displays short messages and/or a progress bar, generally in connection with lengthy operations.
- **Experiment**:
  shows the current experiment, i.e. *Steady-State, Dynamic* or one of the Advanced Experiments
- **Mode**:
  Design or *Run*
- **Zoom**:
  provides a quick access to the same zooming functionality that is included in the Overview Window; one can zoom in or out on the Layout by moving the cursor or clicking on the increment buttons at the two ends of the slider (25% increment steps).

2.8 The Context Menu

The context menu for the Layout Sheet (Figure 2.33) can be accessed by right-clicking anywhere on the canvas.

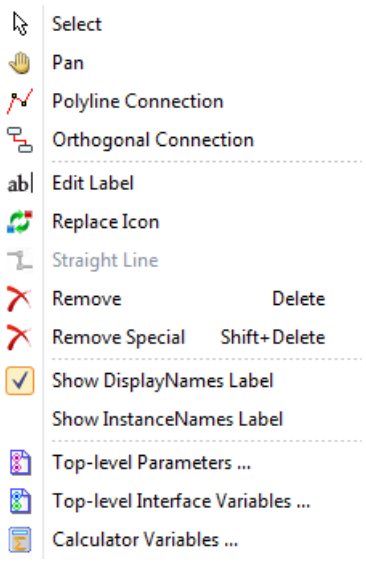


Figure 2.33 Layout Context menu

It comprises the following items:

- **Select**:
  Select an object (cf. Section 2.6.1)
- **Pan**:
  Move the layout thereby focusing on different areas.
- **Polyline Connection**:
  Draw a polygonal connection between two Blocks.
- **Orthogonal Connection**:
  Draw an orthogonal connection between two Blocks.
- **Edit Label** :
  Enabled when a block is selected; to edit the label.
- **Replace Icon**:
  Enabled when a block is selected; to replace the current Block by another Block (existing icon) or by a new generic icon (Figure 2.34).
- **Straight Line**:
  Enabled when a connection is selected; to break the connection in orthogonal segments.
- **Remove**:
  Deletes the selected Block(s) or Connection(s)
- **Remove Special**:
  Works for Blocks only: it removes the Block without deleting its connections, which are instead automatically recreated - if possible
- **Show DisplayNames Label**:
  Show the Display Name as the label for the Blocks.

- **Show InstanceNames Label**:
  Show the Instance Name as the label for the Blocks.
- **Top-level Parameters**:
  Open the dialog (Figure 2.35) to manually create top-level parameters (cf. Section 3.2.3).
- **Top-level Interface Variables**:
  Open the dialog (Figure 2.36) to manually create top-level interface variables (cf. Section 3.2.4).
- **Calculator Variables**:
  Open a dialog window to define a Calculator Variable (cf. Section 3.2.5).

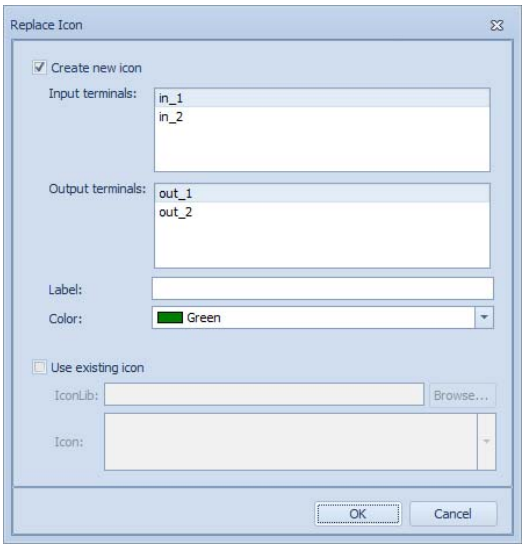


Figure 2.34 Replace Icon dialog window

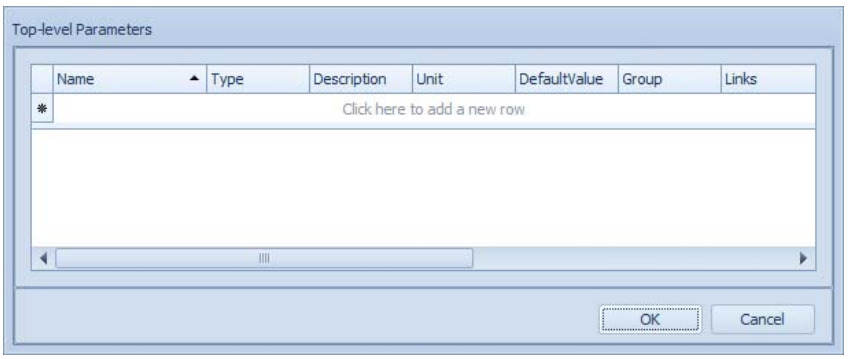


Figure 2.35 Top-level parameters dialog window

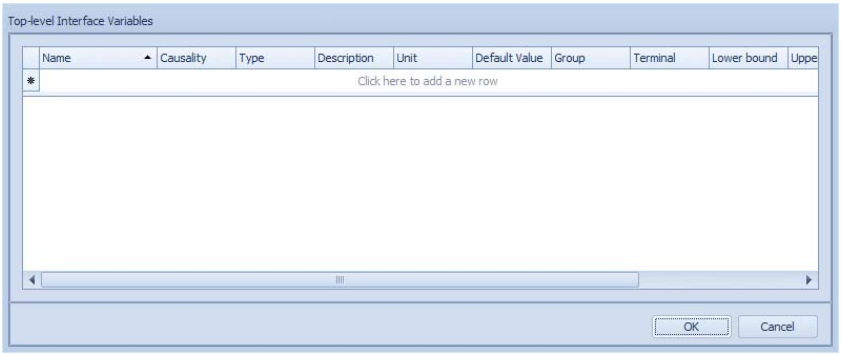


Figure 2.36 Top-level interface variables dialog window

2.9 The Interface Tools

The aim of the Interface Tools is primarily to assist the user in *generating* the input file for the model.

This may be quite a long, tedious, activity since data for the characterization of the influent:

- may come from different sources (hence have different frequency);
- may be only partially available and may therefore need to be complemented with some extra information derived from other sources (e.g. experience or pre-existing knowledge);
- may prove not sufficiently reliable and therefore may need some clean-up to avoid that inaccurate measurements would propagate to the model output.

Note that only the first of the three tasks mentioned above is fully supported in WEST 2020.

Furthermore, determinations of the influent characteristics are typically conducted with reference to *conventional* quantities rather than directly to the specific model components (variables): for instance, COD concentrations rather than S_S, S_I, etc. concentrations are measured. In doing so, there is an inherent incompatibility between influent characteristics expressed in *conventional* units (or: **custom units**) and the units used throughout the WEST model (*model units*, or: **standard units**). An interface is thus needed between the two that translates the custom units into model units. This model is termed: **Fractionation Model**.

To provide an approach consistent with the overall concept of building a model graphically, by arranging blocks and establishing connections between them, the Fractionation Model is to be constructed graphically by means of the Influent Tool.

On the opposite end of the Plant Layout, the characterisation of the effluent is typically expressed with reference to the same custom units that are used to characterise the influent to the plant and as concentrations (for it is to be compared against environmental Standards); so that the transformation opposite to the one carried out on the Input Unit, is to be carried out on the Output Unit, that is a De-fractionation Model (Section 3.3).

The Influent/Effluent Tool can be accessed through a dedicated Sheet that is activated by double-clicking the respective Blocks on the Plant Layout.

2.9.1 Types of Data

Input data can be classified into three functional types:

- component vector
- scalar data
- vector data

The component vector can be specified in two ways, i.e. using the model components (ASM) or using Custom components. The ASM components depend on the Model Instance (ASM1, ASM2d…) selected for the Project. The Custom components are specified by the user and depend on the available measurements. In case Custom components are used a translation to the model components (ASM) needs to be carried out – as explained above.

Scalar data are typically used to manipulate (control) the model or to provide system conditions (e.g. temperature of the water).

Vector data can be used to create vector type interface variables which use the components of one of the Categories of the active Instance.

These functional types of input data correspond to the three classes of input Blocks, i.e. Wastewater, Data- and Vector Data Input respectively (Figure 2.9).

Table 2.3 Overview of the input and output data types

Block	Type	Model/ Interface Var	Data import	(De)Fractionation	Conversion
Influent	Vector	Model	Yes	Yes	C/F
Effluent	Vector	Model	Yes	Yes	F/C
Input	Scalar	Interface Var	Yes	No	No

Table 2.3 Overview of the input and output data types

Block	Type	Model/ Interface Var	Data import	(De)Fractionation	Conversion
Output	Scalar	Interface Var	Yes	No	No
VectorInput	Vector	Interface Var	Yes	No	No
VectorOutput	Vector	Interface Var	Yes	No	No

2.9.2 The Influent Characterisation

The Interface Tool associated with an input Block of type Wastewater ("Municipality", "Industry" or "Truckload") enables the user to do the following:

- Import experimental data from text files and/or spreadsheets.
- Set up a Fractionation Layout that will be turned into a Fractionation Model to convert the measured quantities into the appropriate model variables.
- Revise the Fractionation Layout or the influent file, once the influent model and files have been generated a first time.

The outputs of the Tool are the following:

- A Fractionation Model.
- (Optionally) A conversion model from concentrations and flow rate to loads (fluxes).
- A dynamic influent text file: generated by merging the individual data sources specified by the user.
- A steady-state influent text file: result of the calculation of weighted averages on the dynamic influent file; and used for the steady-state simulation preceding the dynamic simulation.

The Influent Tool Sheet consists of four tab pages ("General", "Fractionation", "Data Import" and "Generate and Review") which represent subsequent *logical* steps in the procedure of generating an input file. Once the input file has been generated for the first time, each of these steps can then be accessed independently to modify specific settings and the input file will have to be regenerated to confirm those changes.

Step 1: General

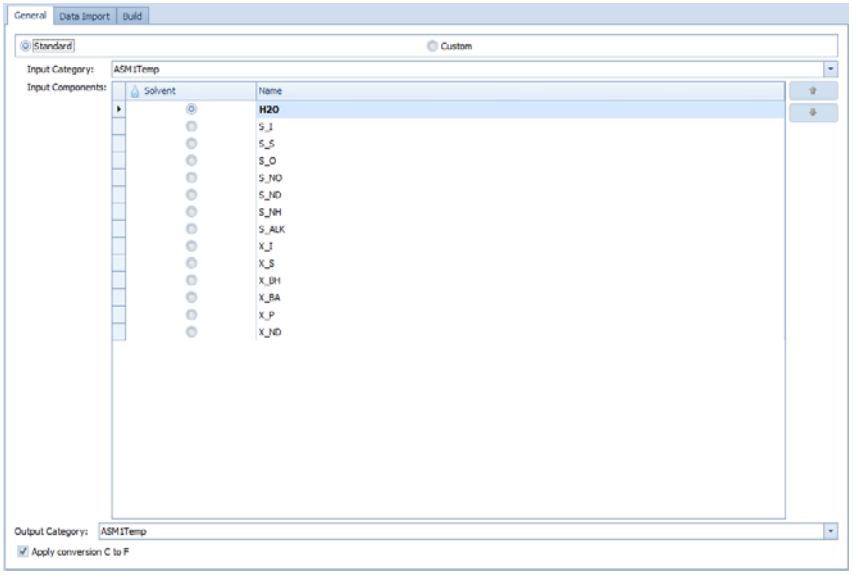


Figure 2.37 General page of the Influent Tool

The first choice one needs to make is whether data are going to be specified using the **Standard** set of category-specific components or a **Custom** set of components.

In the first case (Standard), the **Input Category** denotes the default Category of the model Instance used for the Project. The Input Category determines the component vector shown in the **Input Components** table below.

If Custom is chosen instead, the Input Components table will initially show the conventional set of components, i.e. Water, COD, TKN and TSS. Components can be added to the table and sorted.

Either way, one component needs to be identified as the “solvent” so as to be used to compute weighted averages.

The **Output Category** denotes the default Category of the model Instance used for the Project.

A concentration to flux (C/F) **conversion** is to be added to allow for specifying data as concentrations (except the *solvent*, as flow rate). This option is on by default.

Step 2: Fractionation

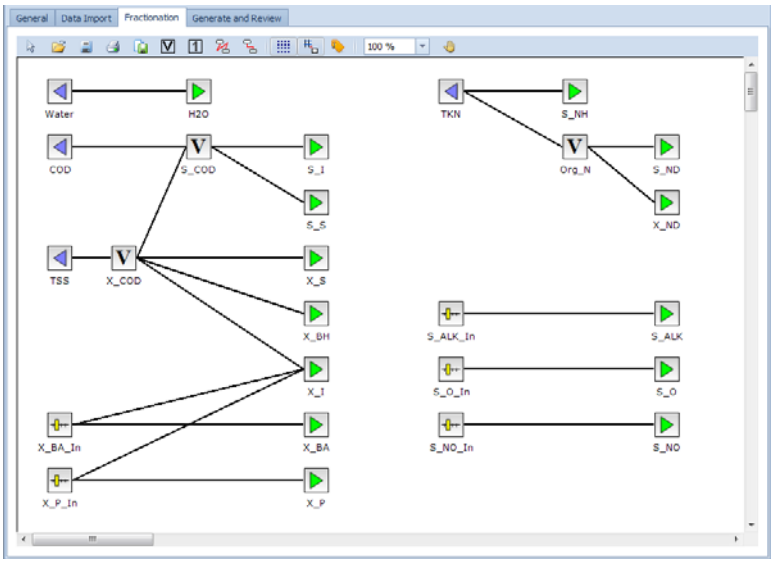


Figure 2.38 Fractionation page of the Influent Tool Sheet

This tab page is not shown if the Standard set of components has been selected in the first page.

This step aims at creating the Fractionation Model i.e. the *interface* between the set of input variables (Custom components) and the set of output variables (the component vector for the selected model Instance/Category).

An existing Fractionation Layout can be imported (Open button in the toolbar) or created graphically by selecting the various classes of Blocks from the toolbar (Input, Output, Parameter or Variable), establishing the appropriate connections (polygonal or orthogonal lines in the toolbar) and assigning a *weight* to every connection (default is 1).

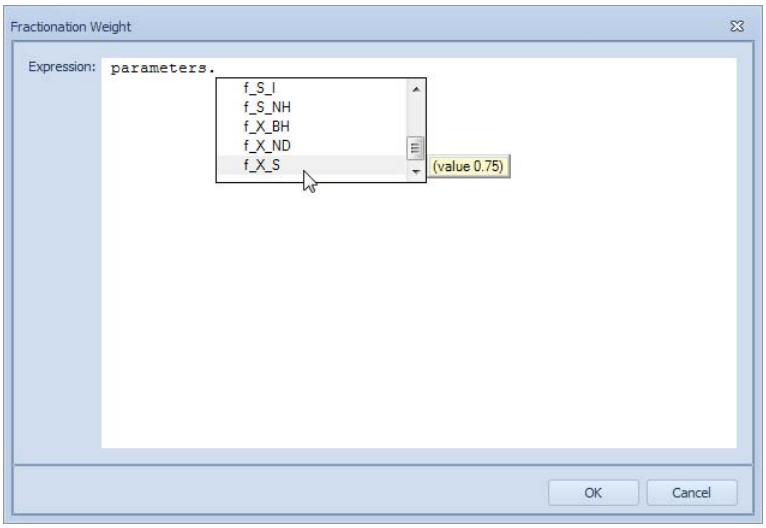


Figure 2.39 Fractionation Weight dialogue window to assign weights to the connections between two blocks

By double-clicking on the connection between two Blocks of the Layout, an expression can be assigned as the weight for that conversion. The usual MSL syntax is supported, i.e. “parameters.<Name>” and “state.<Name>” to use a parameter or a state variable of the model.

Parameters are to be added to the model through the Parameters dialogue, accessible through the context menu (Figure 2.40).

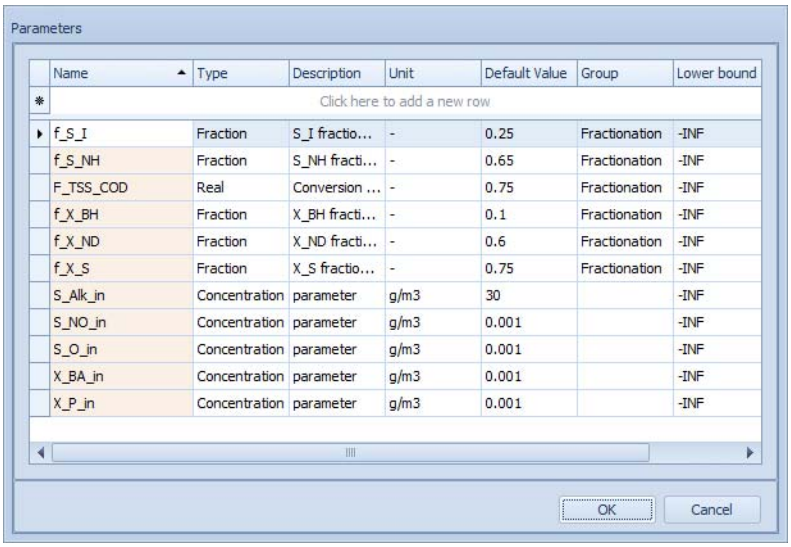


Figure 2.40 Parameters of the Fractionation model

See Section 5.4.5 for a detailed illustration.

Step 3: Data Import

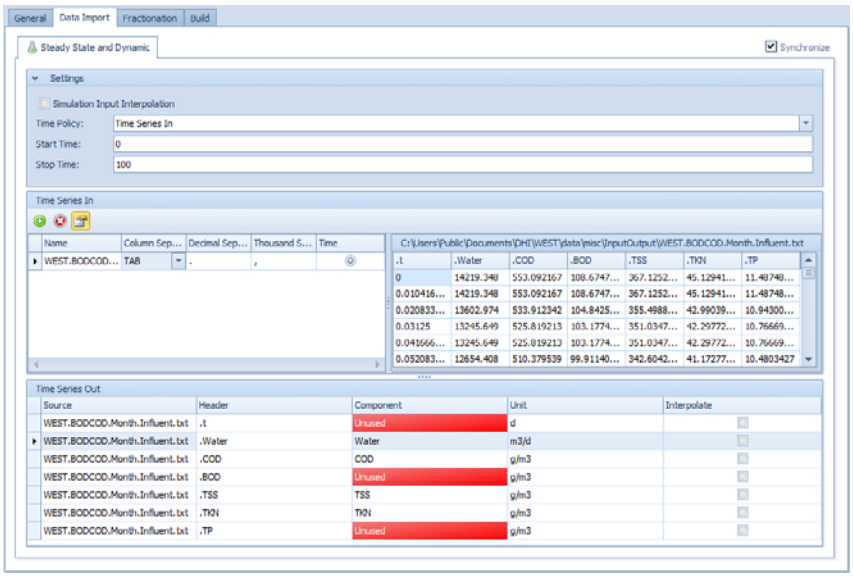


Figure 2.41 Data Import page of the Influent Tool Sheet

The first and general choice one needs to make here is whether data are going to be provided as a dynamic time series (and the steady-state time series is to be computed automatically: Synchronize option, in the top-right corner) or as two separate time series – in which case, two tab subpages will be displayed.

The page (Figure 2.41) consists of three sections: Settings, Time Series In and Time Series Out.

- Settings:
  The following settings are available to generate the input file:
  - **Simulation Input Interpolation**
    If interpolation is on, at every integration time point, the input will be computed by linearly interpolating two consecutive data points: as a result, the simulation speed will generally improve. If interpolation is off, the last available input data point will be maintained.
  - **Time Policy**
    The options are:
    1. Time Series In: uses the time points of one of the source files
    2. Generate
    3. Manual
  - **Start** and **Stop** Time can be specified for all Time Policies.
    In case the “Generate” option is chosen, the number of time points (or the length of the time interval) and the time units are to be further specified.

In case the “Manual” option is chosen, the individual time points and the time units are to be further specified.

- **Time Series In:**
  The grid on the left displays the files that can be used as data source for the composition of the influent file. Text files (.txt) and MS-Excel spreadsheets (.xls, .xlsx) can be read-in provided the correct format is used (cf. Figure 2.42, Figure 2.43 and Figure 2.44).

```
%%Version3.3
%%BeginHeader
t               Water           COD             BOD             TSS
d               m3/d            g/m3            g/m3            g/m3
%%EndHeader
0               14219.348       553.092167      108.6747235     367.1
0.010416667     14219.348       553.092167      108.6747235     367.1
0.020833333     13602.974       533.912342      104.8425113     355.4
0.03125         13245.649       525.819213      103.1774933     351.0
0.041666667     13245.649       525.819213      103.1774932     351.0
0.052083333     12654.408       510.379539      99.91140325     342.6
0.0625          12654.408       510.379539      99.91140325     342.6
0.072916667     12462.7         500.406057      97.606037       336.1
0.083333333     12209.014       477.090316      91.97825825     319.8
0.09375         12209.014       477.090316      91.97825825     319.8
```

Figure 2.42 WEST 3-type input text file

```
#t              Water           COD             BOD             TSS
#d              m3/d            g/m3            g/m3            g/m3
0               14219.348       553.092167      108.6747235     367.1
0.010416667     14219.348       553.092167      108.6747235     367.1
0.020833333     13602.974       533.912342      104.8425113     355.4
0.03125         13245.649       525.819213      103.1774933     351.0
0.041666667     13245.649       525.819213      103.1774932     351.0
0.052083333     12654.408       510.379539      99.91140325     342.6
0.0625          12654.408       510.379539      99.91140325     342.6
0.072916667     12462.7         500.406057      97.606037       336.1
0.083333333     12209.014       477.090316      91.97825825     319.8
0.09375         12209.014       477.090316      91.97825825     319.8
```

Figure 2.43 WEST 2011-type input text file

	A	B	C	D	E	F	G
1	#.t	Water	COD	BOD	TSS	TKN	TP
2	#d	m3/d	g/m3	g/m3	g/m3	g/m3	g/m3
3	0	14219.35	553.0922	108.6747	367.1253	45.12942	11.48749
4	0.010417	14219.35	553.0922	108.6747	367.1253	45.12942	11.48749
5	0.020833	13602.97	533.9123	104.8425	355.4988	42.99039	10.94301
6	0.03125	13245.65	525.8192	103.1775	351.0348	42.29772	10.76669
7	0.041667	13245.65	525.8192	103.1775	351.0348	42.29772	10.76669
8	0.052083	12654.41	510.3795	99.9114	342.6042	41.17277	10.48034
9	0.0625	12654.41	510.3795	99.9114	342.6042	41.17277	10.48034
10	0.072917	12462.7	500.4061	97.60604	336.166	40.5299	10.3167
11	0.083333	12209.01	477.0903	91.97826	319.8166	39.0906	9.950335
12	0.09375	12209.01	477.0903	91.97826	319.8166	39.0906	9.950335

Figure 2.44 WEST 2011-type input Excel spreadsheet

For each file, independent **Column-**, **Decimal**- and **Thousand Separators** can be specified. One file can be flagged as to be used for the reference time scale (**Time** option: only enabled for Time Policy “Time Series In”).

By selecting a file of the list, its content will be displayed in the adjacent grid.

Columns containing the relevant time series are to be dragged into the Time Series Out section to build up the influent file: to select a column, click on its header (column will be highlighted). Multi-selection is possible, using Ctrl (non-consecutive elements) or Shift (consecutive elements) keys.

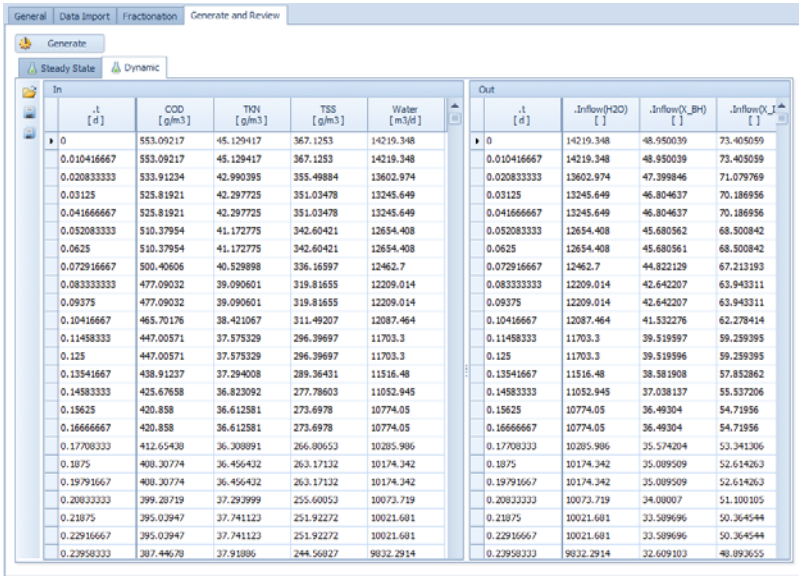
- **Time Series Out:**
This section of the tab page contains a table in which every component of the influent file (either Standard or Custom component as indicated in the General tab page) is to be associated with a specific data series. If the **Header** corresponds to the **Component** name, the association is done automatically; otherwise the user is to manually set the association.

  The following columns are displayed in the grid:

  - **Source**: the source file
  - **Header**: header of the time series
  - **Component**: name of the component
  - **Unit**: it is read from the source file
  - **Interpolate** and **Extrapolate**: it indicates whether interpolation and/or extrapolation is to be carried out on a given data series, while merging.

    Data interpolation at the level of a data series essentially translates to filling up missing data points by linearly interpolating two consecutive data points (interpolation on); or by maintaining the last available data point (interpolation off).

Step 4: Generate and Review



.t [d]	COD [g/m3]	TKN [g/m3]	TSS [g/m3]	Water [m3/d]
0	553.09217	45.129417	367.1253	14219.348
0.010416667	553.09217	45.129417	367.1253	14219.348
0.020833333	533.91234	42.990395	355.49884	13602.974
0.03125	525.81921	42.297725	351.03478	13245.649
0.041666667	525.81921	42.297725	351.03478	13245.649
0.052083333	510.37954	41.172775	342.60421	12654.408
0.0625	510.37954	41.172775	342.60421	12654.408
0.072916667	500.40606	40.529898	336.16597	12462.7
0.083333333	477.09032	39.090601	319.81655	12209.014
0.09375	477.09032	39.090601	319.81655	12209.014
0.10416667	465.70176	38.421067	311.49207	12087.464
0.11458333	447.00571	37.575329	296.39697	11703.3
0.125	447.00571	37.575329	296.39697	11703.3
0.13541667	438.91237	37.294008	289.36431	11516.48
0.14583333	425.67658	36.823092	277.78603	11052.945
0.15625	420.858	36.612581	273.6978	10774.05
0.16666667	420.858	36.612581	273.6978	10774.05
0.17708333	412.65408	36.308891	266.80653	10285.986
0.1875	408.30774	36.456432	263.17132	10174.342
0.19791667	408.30774	36.456432	263.17132	10174.342
0.20833333	399.28719	37.293999	255.60053	10073.719
0.21875	395.03947	37.741123	251.92272	10021.681
0.22916667	395.03947	37.741123	251.92272	10021.681
0.23958333	387.44678	37.91886	244.56827	9832.2914

.t [d]	.Inflow(H2O) [ ]	.Inflow(X_BH) [ ]	.Inflow(X_ [ ]
0	14219.348	48.950039	73.405059
0.010416667	14219.348	48.950039	73.405059
0.020833333	13602.974	47.399846	71.079769
0.03125	13245.649	46.804637	70.186956
0.041666667	13245.649	46.804637	70.186956
0.052083333	12654.408	45.680562	68.500842
0.0625	12654.408	45.680561	68.500842
0.072916667	12462.7	44.822129	67.213193
0.083333333	12209.014	42.642207	63.943311
0.09375	12209.014	42.642207	63.943311
0.10416667	12087.464	41.532276	62.278414
0.11458333	11703.3	39.519597	59.259395
0.125	11703.3	39.519596	59.259395
0.13541667	11516.48	38.581908	57.852862
0.14583333	11052.945	37.038137	55.537206
0.15625	10774.05	36.49304	54.71956
0.16666667	10774.05	36.49304	54.71956
0.17708333	10285.986	35.574204	53.341306
0.1875	10174.342	35.009509	52.514263
0.19791667	10174.342	35.009509	52.514263
0.20833333	10073.719	34.08007	51.100105
0.21875	10021.681	33.509696	50.264544
0.22916667	10021.681	33.509696	50.264544
0.23958333	9832.2914	32.609103	48.899655

Figure 2.45 Generate and Review page of the Influent Tool Sheet (the situation depicted pertains to a custom set of components)

This tab page contains a subpage for the Steady-State and one for the Dynamic input data series.

Each subpage consists of two adjacent tables showing the input (the original data) and the output of the Fractionation Model – if the *Standard* components were selected in the General page, the input and the output coincide.

In order to fill up the tables the first time, the influent files are to be generated by **building** the Fractionation Model. The same build operation is evidently to be performed every time the model is changed (Fractionation step) and/or different data series are used (Data Import step).

The two tables can be edited and the files saved.

Data can be inspected graphically and rescaled, by selecting one or more columns (click on the respective headers) and choosing the corresponding item in the context-menu (right-click): see Figure 2.46.

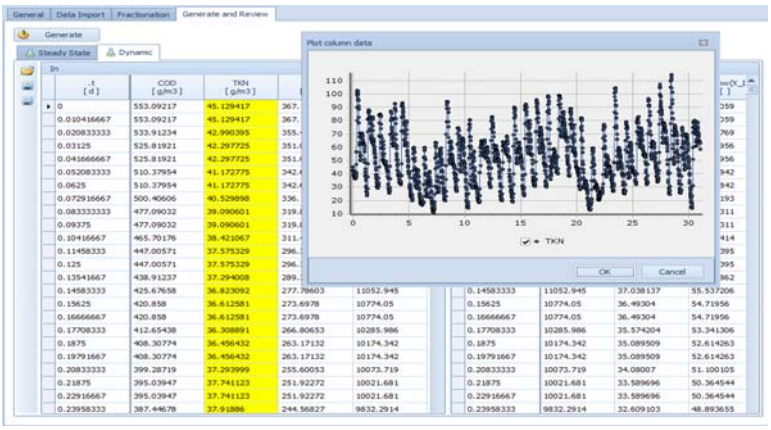


Figure 2.46 Plot of the data series generated by the Influent Tool for visual inspection

2.9.3 Scalar Data Input

The Interface Tool associated with an input Block of type Data is aimed at providing a time series as direct input to a manipulated variable.

Step 1: General

A list of the available scalar top-level interface variables (with “Input” as Causality attribute) is displayed in the “Variables” table (Figure 2.47): select the relevant variable and proceed to “Data Import”.

If the relevant variable is not present in the list (or no scalar input top-level interface variable had been defined), click on the “Top-level Interface Variables” button to access the dialog in Figure 2.36 and proceed as explained in Section 3.2.4.

Step 2: Data Import

See “Data Import” in Section 2.9.2.

Step 3: Generate and Review

See “Generate and Review” in Section 2.9.2. The only difference is that only one Steady-State and one Dynamic file will be generated (and displayed) as no conversion happens through an input model (Figure 2.48).

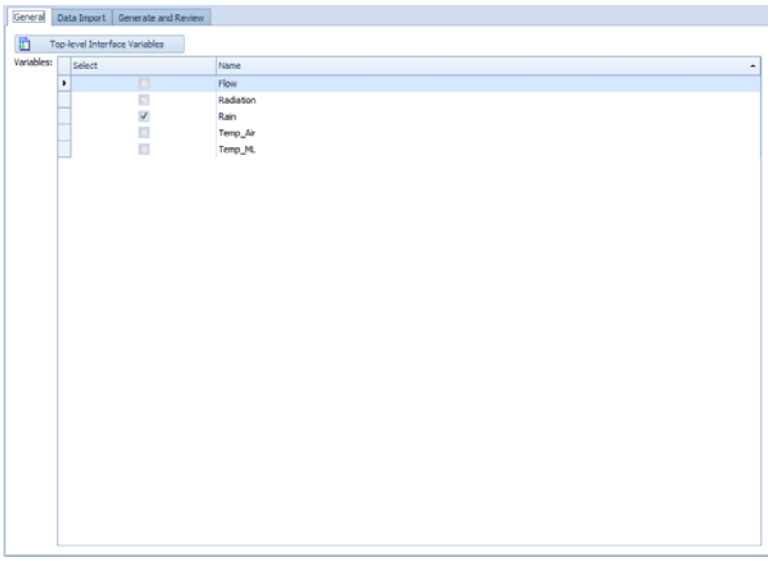


Figure 2.47 General page of the Data Input Tool

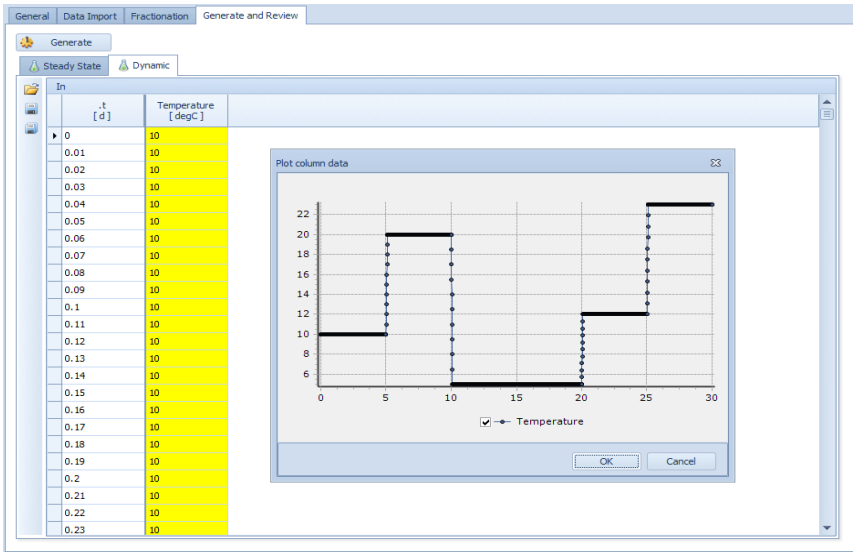


Figure 2.48 Generate and Review page of the Influent Tool Sheet and plot of the time series (the situation depicted pertains to a data input)

2.9.4 Vector Data Input

The Interface Tool associated with an input Block of type Vector Data is aimed at establishing a (top-level) vector interface variable which uses the components of one the Categories of the active Instance.

This is particularly relevant in the scope of the creation of a coupled-model (i.e. hierarchical modelling), because the Vector Input Block also allows for specifying a terminal and interface variable name, and, through the canvas properties, the icon that is to be associated with the sub-layout.

Step 1: General

The Name of the interface variable is to be provided; optionally, the Terminal that the variable will be associated with can be indicated; and a Category, among the ones available for the active Instance, is to be selected - the corresponding Components will be displayed below.

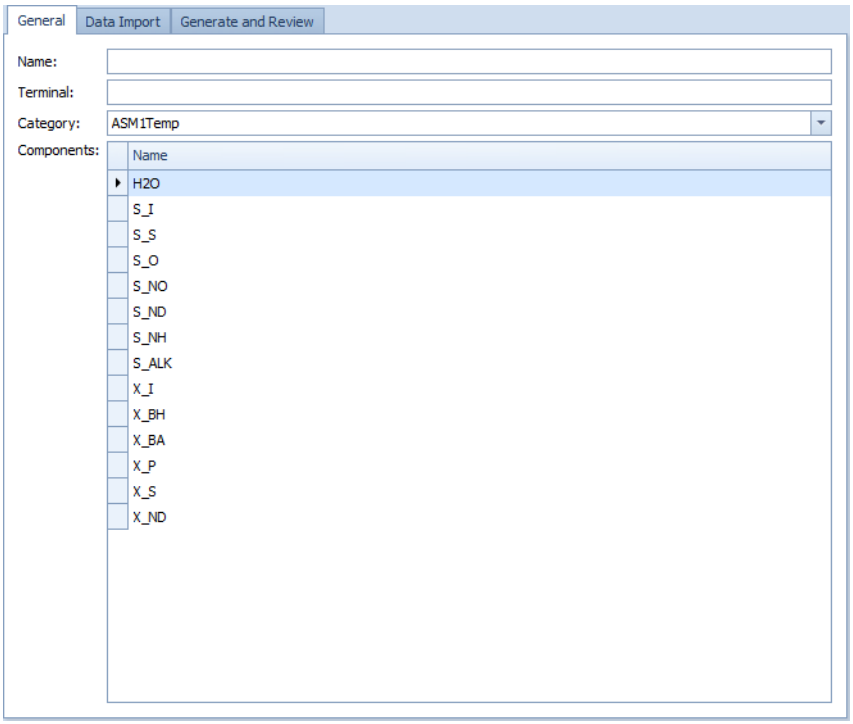


Figure 2.49 Vector data input - General

Step 2: Data Import

See “Data Import” in Section 2.9.2.

Step 3: Generate and Review

See “Generate and Review” in Section 2.9.2. The only difference is that only one Steady-State and one Dynamic file will be generated (and displayed) as no conversion happens through an input model (Figure 2.48).

2.9.5 Merging Data Sources

Data for the characterisation of the influent may originate from different data sources: for instance, the flow rate may be registered by an online flow meter at high frequency (e.g. one value every 5 minutes) whereas COD may be measured offline on daily samples.

As mentioned earlier, the Influent Tool allows for merging different data sources that have different time scales.

Proceed as follows.

- Add the source files in “Times Series In”.
  For each file, specify the Column-, Decimal- and Thousand Separators.
  If the Time Policy in “Settings” is “Time Series In”, select the source file that is to be used as reference for the time scale (column “Time”).
- For each source file, select the columns that contain the relevant times series and drag them to the “Time Series Out” section (to select a column, click on its header - the column will be highlighted).
  Multi-selection is possible, using Ctrl (non-consecutive elements) or Shift (consecutive elements) keys.
- In the “Settings” section, select the appropriate “Time Policy”:
  - Time Series In: to use the time series associated with one of the source files.
  - Generate: to generate the time series by specifying a start and a stop time.
  - Manual: to provide the time points explicitly.

An example of the resulting dynamic influent file that is generated by WEST is shown in the following Figures:

- Figure 2.51 shows a dataset (e.g. online flow rate measurements) which has a frequency greater than or equal to the length of the Time Interval selected in the “Settings” (Time Policy = Generate) – hence data points are sampled by WEST at the appropriate time points
- Figure 2.52 shows a dataset (e.g. offline COD laboratory determinations) which has a frequency smaller than the length of the Time Interval selected in the “Settings” and for which data interpolation is absent – hence WEST takes the available data and holds each value constant until a new value is available at a subsequent time point
- Figure 2.53 shows the same dataset with data interpolation – WEST takes the available data and interpolates **linearly** between two subsequent data points

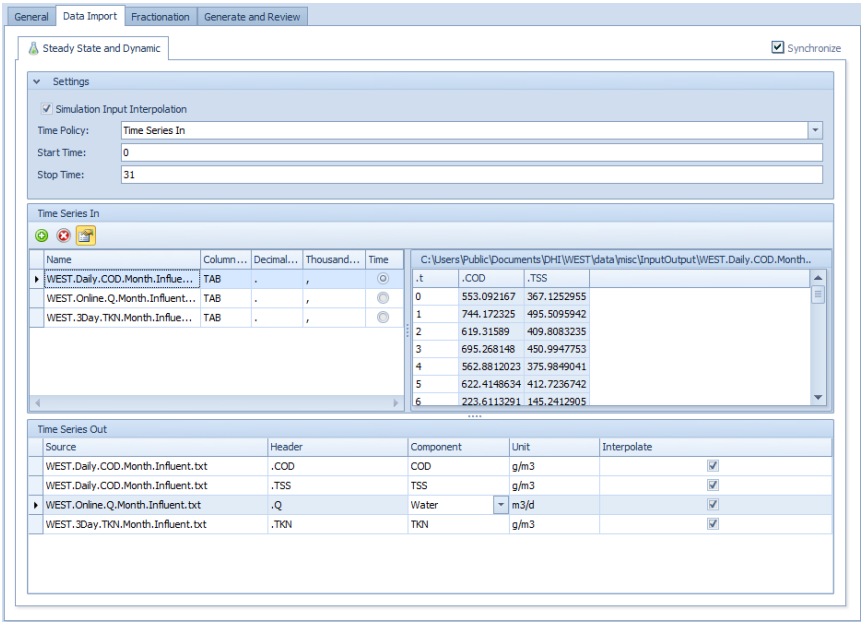


Figure 2.50 Select multiple sources for influent data

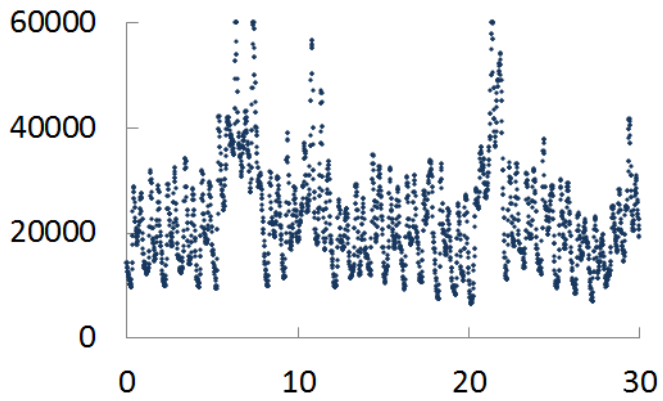


Figure 2.51 Example of dataset at very high frequency

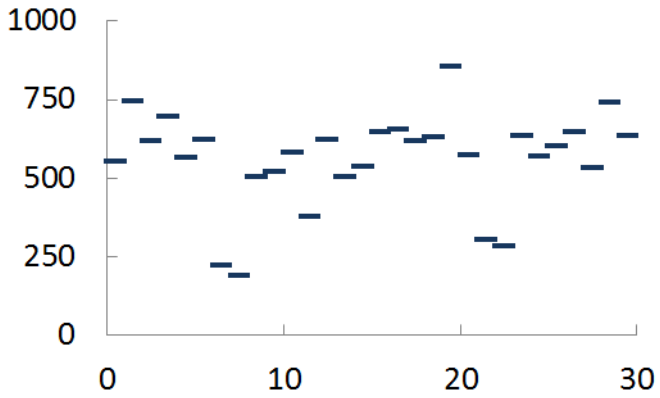


Figure 2.52 Example of dataset at low frequency, without data interpolation

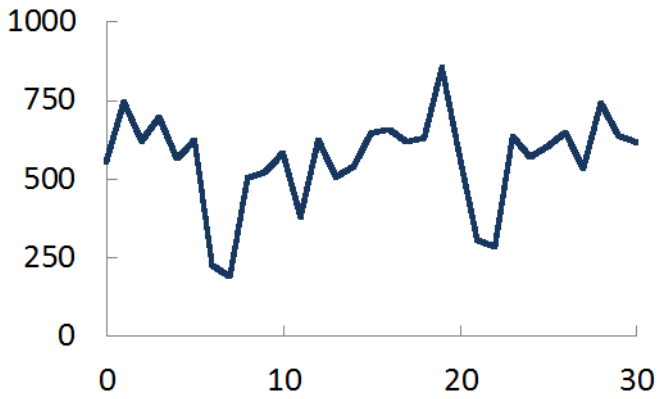


Figure 2.53 Example of dataset at low frequency, with data interpolation

2.9.6 The Effluent Characterisation

The Interface Tool associated with an output Block of type Wastewater (“Effluent” or “Waste”) enables the user to set up (and revise at a later stage) a De-fractionation Layout that will be turned into a De-fractionation Model to convert the model variables into lumped custom variables.

The outputs of the Tool are the De-fractionation Model and (optionally) a conversion model from loads (fluxes) to concentrations and flow.

The Tool consists of three tab pages (“General”, “Defractionation” and “Generate”) which represent subsequent logical steps in the procedure of creating the effluent model. Once this has been generated for the first time, each of these steps can then be accessed independently to modify specific settings and the model will have to be regenerated to confirm those changes.

Step 1: General

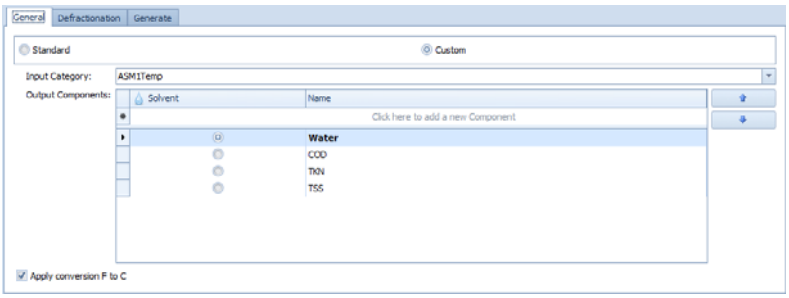


Figure 2.54 General page of the Effluent Tool

The first choice one needs to make is whether the effluent is to be specified using the Standard set of category-specific components or a Custom set of components.

In the first case (Standard), the Input Category denotes the default Category of the model Instance used for the Project. The Input Category determines the component vector shown in the Input Components table below.

If Custom is chosen instead, the Output Components table will initially show the conventional set of components, i.e. Water, COD, TKN and TSS. Components can be added to the table and sorted.

Either way, one component needs to be identified as the “solvent” so as to be used to compute weighted averages.

The Output Category denotes the default Category of the model Instance used for the Project.

A flux to concentration (F/C) conversion is to be added to allow for specifying effluent as concentrations (except the solvent, as flow rate). This option is on by default.

Step 2: Defractionation

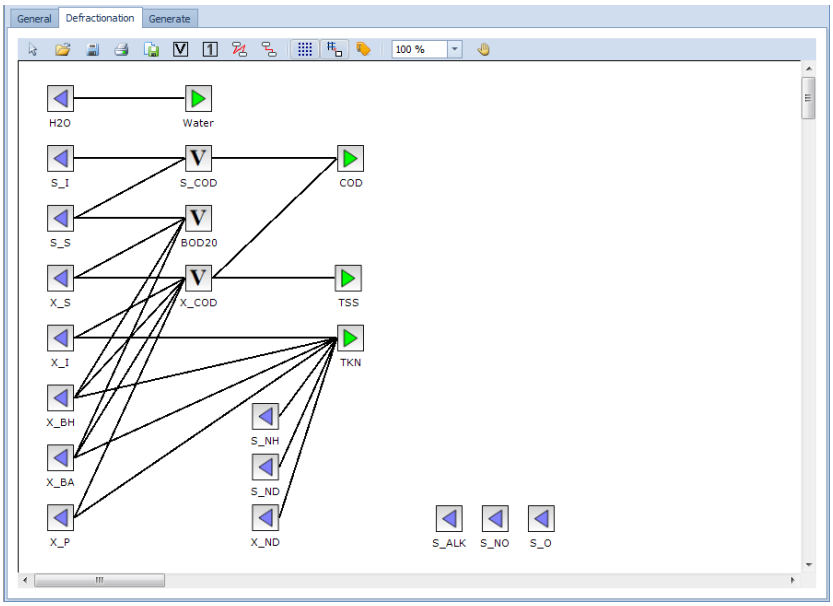


Figure 2.55 Defractionation page of the Effluent Tool Sheet

This tab page is not shown if the Standard set of components has been selected in the first page.

This step aims at creating the De-fractionation Model i.e. the interface between the set of input variables (the component vector for the selected model Instance/Category) and the set of output variables (Custom components).

An existing De-fractionation Layout can be imported (Open button in the toolbar) or created graphically by selecting the various classes of Blocks from the toolbar (Input, Output, Parameter or Variable), establishing the appropriate connections (polygonal or orthogonal lines in the toolbar) and assigning a weight to every connection (default is 1).

By double-clicking on the connection between two Blocks of the Layout, an expression can be assigned as the weight for that conversion. The usual MSL syntax is supported, i.e. “parameters.<Name>” and “state.<Name>” to use a parameter or a state variable of the model.

Parameters are to be added to the model through the Parameters dialogue, accessible through the context menu (Figure 2.40).

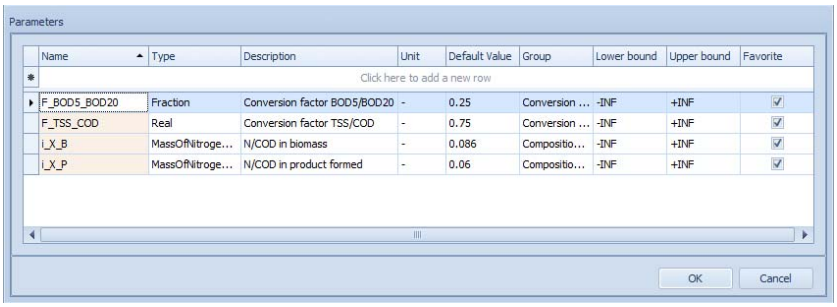


Figure 2.56 Parameters of the defractionation model

Step 3: Generate

To effectively create the De-fractionation model (or to regenerate it, every time its Layout has been modified), click on the “Generate” button.

2.9.7 Scalar Data Output

The Interface Tool associated with an output Block of type Data serves only the purpose of exposing a scalar top-level interface variable.
Only one tab page is therefore present.

General

Select the relevant variable, in the list of available scalar top-level interface variables (with “Output” as Causality attribute), displayed in the “Variables” table (Figure 2.46).

If the variable is not present in the list (or no output top-level interface variable had been defined), click on the “Top-level Interface Variables” button to access the dialog in Figure 2.36 and proceed as explained in Section 3.2.4.

2.9.8 Vector Data Output

The Interface Tool associated with an input Block of type Vector Data serves only the purpose of exposing a top-level vector interface variable.
Only one tab page is therefore present.

This is particularly relevant in the scope of the creation of a coupled-model (i.e. hierarchical modelling), because the Vector Output Block also allows for specifying a terminal and interface variable name, and, through the canvas properties, the icon that is to be associated with the sub-layout.

General

The Name of the interface variable is to be provided; optionally, the Terminal that the variable will be associated to can be indicated; and a Category,

among the ones available for the active Instance, is to be selected - the corresponding Components will be displayed below (Figure 2.49).

2.10 The Properties of the GUI Objects

As mentioned earlier, the Properties window is a versatile grid that exposes the properties of the object that is currently selected on the Layout Sheet (i.e. a Block or another element of the Layout) or on a Dashboard Sheet.

2.10.1 Properties of the Layout Sheet

The main properties of the Layout Sheet are:

- Description:
  Description of the coupled-model associated with the Layout.
- Icon Name:
  Icon that is to be associated to the coupled-model.
- IsDefault:
  whether the coupled-model is the default for the selected icon.
- **Line Bridging**:
  line bridging is applied when two connections intersect. Default: False. The same property is available at the level of the individual connection.
- **Line Routing**:
  if True and in combination with the TreatAsObstacle property of a Block also set to True, if the connection is drawn over a Block, it will be automatically re-drawn to avoid it. Default: False. The same property is available at the level of the individual connection.
- **InsertLoopBreakers**:
  whether algebraic loop-breakers are to be inserted automatically.
- **LoopBreaker**:
  type of loop-breaker to be inserted - currently, the only type available is the differential loop-breaker.

2.10.2 Properties of a Block

A Block consists of two elements, i.e. an Icon and a Label.

The Block as a whole as well as its elements have specific properties.

The following Properties of the Block are visualised in the Properties window by a single mouse click on the Block (Figure 2.57, left).

- **PinPoint Offset**:
  position of the pin point (the centre of the node) (read-only).

- **Size**:
  size of the Block in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Class Name**:
  the model associated with the Block. The dropdown allows for selecting a different model, if multiple models should be available.
- **Data**:
  set of read-only data.
- **Description**:
  description.
- **Display Name**:
  name of the block that is to be displayed on the layout.
- **Icon Name**:
  name of the block icon (read-only).
- **Name**:
  (internal) name of the Block - it is displayed e.g. in the Model Explorer.
- **ShowLabel**:
  attribute to show or hide the label (True, by default).
- **Synchronise Names**:
  to couple (True) or decouple (False) the Display Name and the Name.
- **Layers**:
  layer to which the Block belongs.
- **TreatAsObstacle**:
  if True (default) and in combination with the LineRouting property of a connection also set to True, the Block is treated as an obstacle, i.e. a connection that is drawn over it, will be automatically re-drawn to avoid the Block.
- **Flip X**:
  attribute showing the Horizontal Flip status of the Block.
- **Flip Y**:
  attribute showing the Vertical Flip status of the Block.
- **Rotation Angle**:
  attribute showing the rotation angle of the Block.

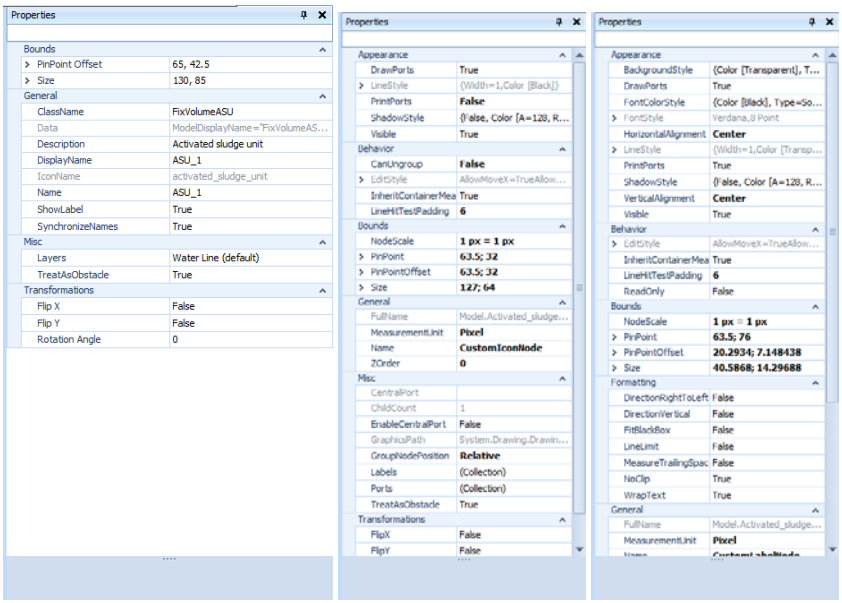


Figure 2.57 Properties of a Block (left), Block Icon (centre) and Block Label (right)

Properties of the Block Icon

The Properties of the Block Icon are visualised in the Properties window by clicking first on the Block and a second time on its Icon (Figure 2.57, centre).

The main Properties are:

- **LineStyle**:
  denotes the aggregate of several properties of the pen used to draw the lines, such as DashStyle, LineColor, LineWidth, etc.
- **Visible**:
  attribute to make the Block Icon invisible (False); True, by default.
- **PinPoint**:
  denotes the anchor point for the Block Icon that identifies its position in the Layout.
- **Size**:
  indicates the size of the Block Icon in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Name**:
  name of the Block Icon.
- **Flip X**:
  if set to True, the Block Icon is mirrored on its vertical axis (False, by default).

- **Flip Y**:
  if set to True, the Block Icon is mirrored on its horizontal axis (False, by default).
- **Rotation Angle**:
  indicates the angle of rotation.

Properties of the Block Label

The following Properties of the Block Label are visualised in the Properties window by clicking first on the Block and a second time on its Label (Figure 2.57, right).

The main Properties are:

- **FontStyle**:
  collection of properties related to the font style.
- **HorizontalAlignment**:
  horizontal alignment of the text
- **LineStyle**:
  the aggregate of several properties of the pen used to draw the lines, such as DashStyle, LineColor, LineWidth, etc.
- **VerticalAlignment**:
  horizontal alignment of the text
- **Visible**:
  attribute to make the Block Label invisible (False); True, by default.
- **PinPoint**:
  denotes the anchor point for the Block Label that identifies its position in the Layout.
- **Size**:
  indicates the size of the Block Label in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Name**:
  of the Block Label.
- **Text**:
  text displayed in the Label.
- **Flip X**:
  if set to True, the Block Icon is mirrored on its vertical axis (False, by default).
- **Flip Y**:
  if set to True, the Block Icon is mirrored on its horizontal axis (False, by default).
- **Rotation Angle**:
  indicates the angle of rotation.

2.10.3 Properties of a Rectangle

The main Properties of a Rectangle object are:

- **FillStyle**:
  colour and style of the filling.
- **LineStyle**:
  denotes the aggregate of several properties of the pen used to draw the lines, such as DashStyle, LineColor, LineWidth, etc.
- **Visible**:
  attribute to make the object invisible (False); True, by default.
- **PinPoint**:
  denotes the anchor point for the object that identifies its position in the Layout.
- **Size**:
  indicates the size of the object in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Name**:
  name of the object.
- **Flip X**:
  if set to True, the object is mirrored on its vertical axis (False, by default).
- **Flip Y**:
  if set to True, the object is mirrored on its horizontal axis (False, by default).
- **Rotation Angle**:
  indicates the angle of rotation.

2.10.4 Properties of a Textbox

The main Properties of a Textbox object are:

- **FontColorStyle**:
  colour and style of the text.
- **FontStyle**:
  collection of properties related to the font style.
- **HorizontalAlignment**:
  horizontal alignment of the text
- **LineStyle**:
  denotes the aggregate of several properties of the pen used to draw the lines, such as DashStyle, LineColor, LineWidth, etc.
- **VerticalAlignment**:
  horizontal alignment of the text

- **Visible**:
  attribute to make the object invisible (False); True, by default.
- **PinPoint**:
  denotes the anchor point for the object that identifies its position in the Layout.
- **Size**:
  indicates the size of the object in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Name**:
  name of the object.
- **Text**:
  text displayed.
- **Flip X**:
  if set to True, the object is mirrored on its vertical axis (default: False).
- **Flip Y**:
  if set to True, the object is mirrored on its horizontal axis (default: False).
- **Rotation Angle**:
  indicates the angle of rotation.

2.10.5 Properties of a Picture

The main Properties of a Picture object are:

- **LineStyle**:
  denotes the aggregate of several properties of the pen used to draw the lines, such as DashStyle, LineColor, LineWidth, etc.
- **ShadowStyle**:
  denotes the aggregate of several properties of the shadow around the object area
- **Visible**:
  attribute to make the object invisible (False); True, by default.
- **PinPoint**:
  denotes the anchor point for the object that identifies its position in the Layout.
- **Size**:
  indicates the size of the object in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Name**:
  name of the object
- **Image**:
  link to the image associated to the object (and browse dialog)

- **Flip X**:
  if set to True, the object is mirrored on its vertical axis (default: False).
- **Flip Y**:
  if set to True, the object is mirrored on its horizontal axis (default: False).
- **Rotation Angle**:
  indicates the angle of rotation

2.10.6 Properties of a Line

The main Properties of a Line (Pencil) object are:

- **LineStyle**:
  denotes the aggregate of several properties of the pen used to draw the lines, such as DashStyle, LineColor, LineWidth, etc.
- **ShadowStyle**:
  denotes the aggregate of several properties of the shadow around the object area
- **Visible**:
  attribute to make the object invisible (False); True, by default.
- **PinPoint**:
  denotes the anchor point for the object that identifies its position in the Layout.
- **Size**:
  indicates the size of the object in pixels; it can be modified in line or through its individual components, i.e. Width and Height.
- **Name**:
  name of the object
- **Flip X**:
  if set to True, the object is mirrored on its vertical axis (default: False).
- **Flip Y**:
  if set to True, the object is mirrored on its horizontal axis (default: False).
- **Rotation Angle**:
  indicates the angle of rotation

2.10.7 Properties of a Connection

The main properties of a Connection are:

- **Layers**:
  the layers that the Connection belongs to
- **Line Bridging**:
  line bridging is applied when two connections intersect. Default: False.

- **Line Routing**:
  if True and in combination with the TreatAsObstacle property of a Block also set to True, if the connection is drawn over a Block, it will be automatically re-drawn to avoid it. Default: False.

2.10.8 Properties of a Dashboard Sheet

The only Property of a Dashboard Sheet is its Display Name.

2.10.9 Properties of a Slider Pane

A Slider pane acts as a container (or sub-container, since the Sheet is the overall container for Dashboard widgets) that can host several parameters although each one will be assigned to its individual Parameter Slider.

A Slider Pane has the following Properties:

- **Display Name**:
  name of the Slider pane (as the container).
- **ShowCaption**:
  whether the caption is to be displayed above the pane.

A sub-menu is displayed by clicking on blue arrow in the upper right corner of the Slider pane and gives access to the following items (Figure 2.58):

- Add Slider:
  allows for adding a slider - alternative to drag-drop from the Block Details or Block Summary panes
- **Slider Pane Properties**:
  shows the properties of the Slider Pane, i.e. its Name.
- **Slider Properties**:
  further visualises a sub-menu containing the list of the Parameters Sliders.
- **Remove Slider Pane**:
  deletes the Slider pane from the Sheet.
- **Remove Slider**:
  further visualises a sub-menu containing the list of Parameter Sliders that can be individually deleted from the Slider pane.
- **Design Mode**:
  allows for arranging the individual sliders within the pane.

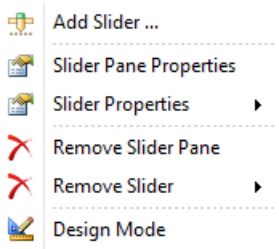


Figure 2.58 Sub-menu for the Slider pane

Properties of a Parameter Slider

The following Properties of a Parameter Slider are visualised in the Properties window by clicking on the respective item of the sub-menu of a Slider Pane (Figure 2.59).

The main Properties are:

- **Description**:
  free text field in which an informative description of the quantity can be inputted.
- **Name**:
  name that is assigned to the slider; by default, the parameter name, but can be renamed for convenience.
- **Quantities**:
  this is the key property as it denotes the model parameter that is represented through the slider; it is assigned by dragging the parameter from the Block Details window onto the Slider.
- **Unit**:
  (read-only) default unit for the parameter, as it is specified in the model library.
- **Visible**:
  attribute to turn on (True, default) or off (False) the slider.
- **Frequency**:
  distance between two adjacent divisions on the slider.
- **LargeChange**:
  step changes when using the “PageUp” or “PageDown” keys.
- **SmallChange**:
  step changes when using the arrows keys.

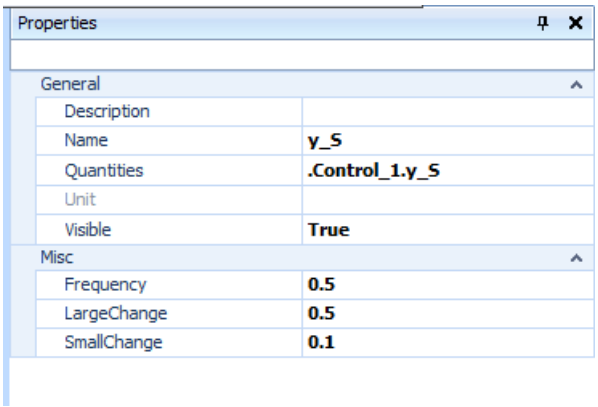


Figure 2.59 Properties of a Parameter Slider

2.10.10 Properties of an Input (Output) Field Pane

An Input (or Output) Field pane acts as a *container* (or sub-container, since the Sheet is the overall container for Dashboard widgets) that can host several parameters or variables although each one will be assigned to its individual Field.

The **main** Properties of an Input (Output) Field pane are:

- **Name**:
  name of the Field pane (as the container).
- **ShowCaption**:
  whether the caption is to be displayed above the pane.

The difference between an Input - and an Output Field is that the former only accepts parameters, as it will enable to manipulate the value during the simulation, whereas the Output Field accepts both parameters and variables, as it displays the current value.

A sub-menu is displayed by clicking on blue arrow in the upper right corner of the Field pane and gives access to the following items (Figure 2.60):

- Add Field:
  allows for adding a field - alternative to drag-drop from the Block Details or Block Summary panes.
- **Field Pane Properties**:
  shows the properties of the Field Pane, i.e. its Name.
- **Field Properties**:
  further visualises a sub-menu containing the list of the individual Fields.
- **Remove Field Pane**:
  deletes the Field pane from the Sheet.

- **Remove Field**:
  further visualises a sub-menu containing the list of individual Fields that can be individually deleted from the Field pane.
- **Design Mode**:
  allows for arranging the individual fields within the pane.

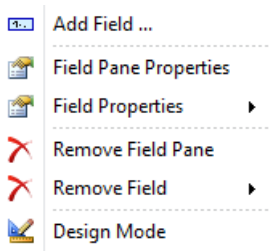


Figure 2.60 Sub-menu for an Input (Output) Field pane

Properties of an Input (Output) Field

The following Properties of an Input or Output Field are visualised in the Properties window by clicking on the respective item of the Context Menu of a Field Pane (Figure 2.61).

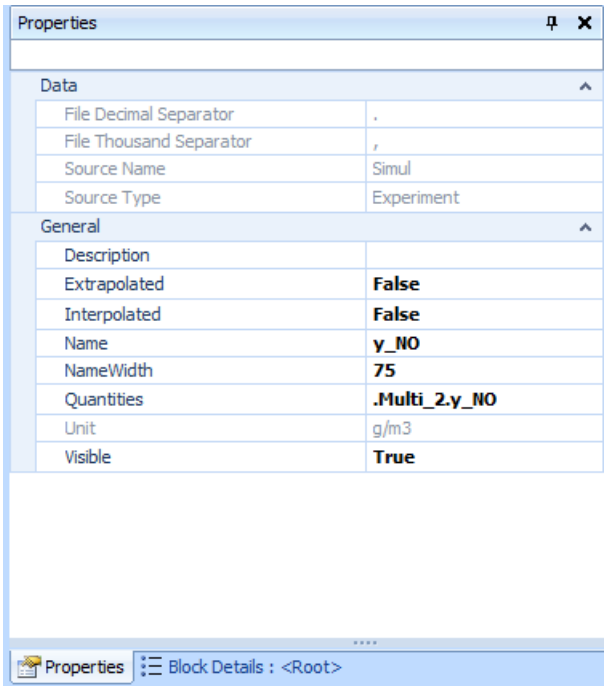


Figure 2.61 Properties of an Input (Output) Field

The **main** Properties are:

- **Description**:
  free text field in which an informative description of the quantity can be inputted.
- Extrapolated
- Interpolated
- **Name**:
  name that is assigned to the field; by default, the parameter or variable name, but can be renamed for convenience.
- NameWidth:
  the width of the label.
- **Quantities**:
  this is the key property as it denotes the model parameter or variable that is represented through the Field; it is assigned by dragging the parameter or variable from the Process Details window onto the Field pane.
- **Unit**:
  default unit for the parameter or variable, as it is specified in the model library.
- **Visible**:
  attribute to turn on (True, default) or off (False) the Field.

2.10.11 Properties of a Radiobutton Pane

A Radiobutton pane acts as a *container* (or sub-container, since the Sheet is the overall container for Dashboard widgets) that can host several parameters although each one will be assigned to its individual Radiobutton.

The **main** Properties of a Radiobutton pane are:

- **Name**:
  name of the Radiobutton pane (as the container).
- **ShowCaption**:
  whether the caption is to be displayed above the pane.

A sub-menu is displayed by clicking on blue arrow in the upper right corner of the Radiobutton pane and gives access to the following items (Figure 2.62):

- Add Radiobutton:
  allows for adding a radiobutton - alternative to drag-drop from the Block Details or Block Summary panes.
- **Radiobutton Pane Properties**:
  shows the properties of the Radiobutton Pane, i.e. its Name.

- **Radiobutton Properties**:
  further visualises a sub-menu containing the list of the individual Radiobuttons.
- **Remove Radiobutton Pane**:
  deletes the Radiobutton pane from the Sheet.
- **Remove Radiobutton**:
  further visualises a sub-menu containing the list of individual Radiobuttons that can be individually deleted from the Radiobutton pane.
- **Design Mode**:
  allows for arranging the individual radiobuttons within the pane.

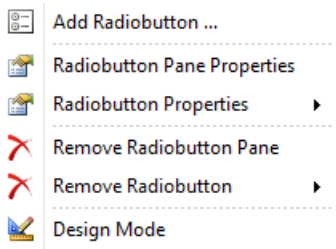


Figure 2.62 Sub-menu for a Radiobutton pane

Properties of a Radiobutton

The following Properties of a Radiobutton are visualised in the Properties window by clicking on the respective item of the Context Menu of a Radiobutton Pane (Figure 2.63).

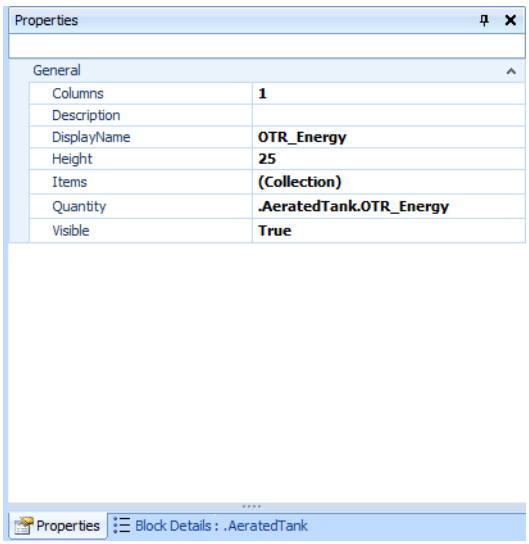


Figure 2.63 Properties of a Radiobutton

The **main** Properties are:

- Columns:
  number of columns that are to be used to display the individual radiobuttons
- **Description**:
  free text field in which an informative description of the quantity can be inputted.
- **DisplayName**:
  name that is assigned to the radiobutton; by default, the parameter name, but can be renamed for convenience.
- Height:
  the height of the radiobutton.
- Items:
  exposes the RadioGroupItem Collection Editor which allows for adding, sorting and deleting individual items
- **Quantity**:
  this is the key property as it denotes the model parameter that is represented through the Radiobutton; it is assigned by dragging the parameter from the Process Details window onto the Radiobutton pane.
- **Visible**:
  attribute to turn on (True, default) or off (False) the Radiobutton.

2.10.12 Properties of a Combobox Pane

A Combobox pane acts as a *container* (or sub-container, since the Sheet is the overall container for Dashboard widgets) that can host several parameters although each one will be assigned to its individual Combobox.

The **main** Properties of a Combobox pane are:

- **Name**:
  name of the Combobox pane (as the container).
- **ShowCaption**:
  whether the caption is to be displayed above the pane.

A sub-menu is displayed by clicking on blue arrow in the upper right corner of the Combobox pane and gives access to the following items (Figure 2.64):

- Add Combobox:
  allows for adding a combobox - alternative to drag-drop from the Block Details or Block Summary panes.
- **Combobox Pane Properties**:
  shows the properties of the Combobox Pane, i.e. its Name.

- **Combobox Properties**:
  further visualises a sub-menu containing the list of the individual Comboboxes.
- **Remove Combobox Pane**:
  deletes the Combobox pane from the Sheet.
- **Remove Combobox**:
  further visualises a sub-menu containing the list of individual Comboboxes that can be individually deleted from the Combobox pane.
- **Design Mode**:
  allows for arranging the individual comboboxes within the pane.

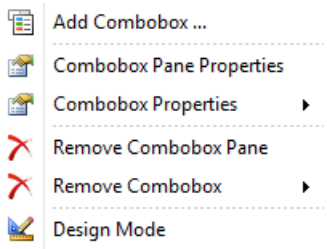


Figure 2.64 Sub-menu for a Combox pane

Properties of a Combobox

The following Properties of a Combobox are visualised in the Properties window by clicking on the respective item of the Context Menu of a Combobox Pane (Figure 2.65).

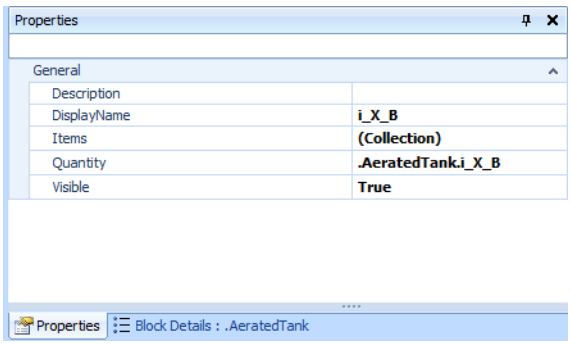


Figure 2.65 Properties of a Combobox

The **main** Properties are:

- **Description**:
  free text field in which an informative description of the quantity can be inputted.

- **DisplayName**:
  name that is assigned to the combobox; by default, the parameter name, but can be renamed for convenience.
- Items:
  exposes the ComboboxItem Collection Editor which allows for adding, sorting and deleting individual items
- **Quantity**:
  this is the key property as it denotes the model parameter that is represented through the Combobox; it is assigned by dragging the parameter from the Process Details window onto the Combobox pane.
- **Visible**:
  attribute to turn on (True, default) or off (False) the Combobox.

2.10.13 Properties of a Checkbox Pane

A Checkbox pane acts as a *container* (or sub-container, since the Sheet is the overall container for Dashboard widgets) that can host several parameters although each one will be assigned to its individual Checkbox.

The **main** Properties of a Checkbox pane are:

- **Name**:
  name of the Checkbox pane (as the container).
- **ShowCaption**:
  whether the caption is to be displayed above the pane.

A sub-menu is displayed by clicking on blue arrow in the upper right corner of the Checkbox pane and gives access to the following items (Figure 2.66):

- Add Checkbox:
  allows for adding a checkbox - alternative to drag-drop from the Block Details or Block Summary panes.
- **Checkbox Pane Properties**:
  shows the properties of the Checkbox Pane, i.e. its Name.
- **Checkbox Properties**:
  further visualises a sub-menu containing the list of the individual Checkboxes.
- **Remove Checkbox Pane**:
  deletes the Radiobutton pane from the Sheet.
- **Remove Checkbox**:
  further visualises a sub-menu containing the list of individual Checkboxes that can be individually deleted from the Checkbox pane.
- **Design Mode**:
  allows for arranging the individual checkboxes within the pane.

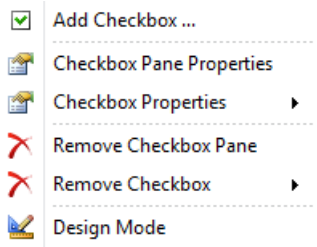


Figure 2.66 Sub-menu for a Checkbox pane

Properties of a Checkbox

The following Properties of a Checkbox are visualised in the Properties window by clicking on the respective item of the Context Menu of a Checkbox Pane (Figure 2.67).

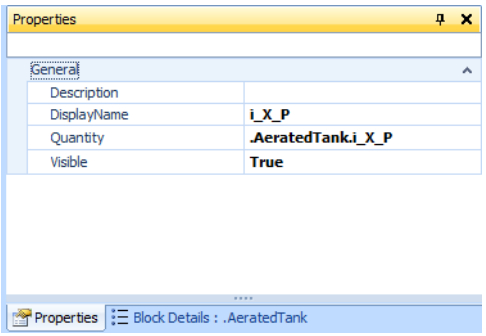


Figure 2.67 Properties of a Checkbox

The **main** Properties are:

- **Description**:
  free text field in which an informative description of the quantity can be inputted.
- **DisplayName**:
  name that is assigned to the checkbox; by default, the parameter name, but can be renamed for convenience.
- **Quantity**:
  this is the key property as it denotes the model parameter that is represented through the Checkbox; it is assigned by dragging the parameter from the Process Details window onto the Checkbox pane.
- **Visible**:
  attribute to turn on (True, default) or off (False) the Checkbox.

2.10.14 Properties of a Plot Window

A Context Menu is displayed by clicking on blue arrow in the upper right corner of the Plot window which gives access to the following items (cf. Figure 2.68):

- **Add Series**:
  opens up the 'Add Series' dialog window, illustrated in Section 5.7.3.
- **Arrange Series**:
  allows for sorting the data series.
- **Plot Properties**:
  displays the properties of the Plot window in the Properties pane (see below).
- **Series Properties**:
  displays the properties of the selected Data Series in the Properties pane (see below).
- **Axes Properties**:
  displays the properties of the selected Axis in the Properties pane (see below).
- **Legend Properties**:
  displays the properties of the Legend in the Properties pane (see below).
- Header:
  displays the properties of the Header in the Properties pane
- Footer:
  displays the properties of the Footer in the Properties pane
- **Export**:
  exports the plot to a variety of graphic formats.
- **Clear All Data Series**:
  clears all data series.
- **Refresh Data Series**:
  refreshes all data series.
- **Remove Plot**:
  removes the Plot window.
- **Remove Series**:
  removes the selected (or all) Data Series.
- **Rotate**:
  rotates the chart (only for 3D plots).
- **Crosshair**:
  displays a crosshair and the coordinates of the point.
- **Series Bands**:
  allows for colouring the area below a time series (i.e. its integral).

- **Scroll Pager**:
  when activated, the scroll pager appears below the actual plot pane and can be used to move a zoomed-in “window” across the time horizon.
- **Copy Image to Clipboard**:
  copies the image to the clipboard

The **main** Properties of a Plot window object are:

- **Aspect**:
  properties related to the general aspect of the chart.
- **Background**:
  background style.
- **Gradient**:
  properties of the gradient, e.g. Visible (False, by default).
- **Walls**:
  properties of the graph walls, e.g. Visible (False, by default).
- **DataSet**:
  type of dataset, e.g. Time series or Current Value (read-only).
- **Name**:
  name of the Plot Window.
- **Type**:
  type of chart, e.g. Line, Bar, … (read-only).
- **Visible**:
  indicates whether the Plot Window is to be visible or not on the Sheet.
- **Axes**:
  for each axis, several sub-properties can be set: see ‘Properties of the Axes’.
- **Legend**:
  several sub-properties can be set: see ‘Properties of the LegendProperties of the Legend’.
- **Footer**:
  title of the Plot Window that appears below the chart area; the usual properties of a text object, e.g. Alignment, Font, Font Size, etc. are available as sub-properties of the Footer.
- **Header**:
  title of the Plot Window that appears above the chart area; the usual properties of a text object, e.g. Alignment, Font, Font Size, etc. are available as sub-properties of the Footer.

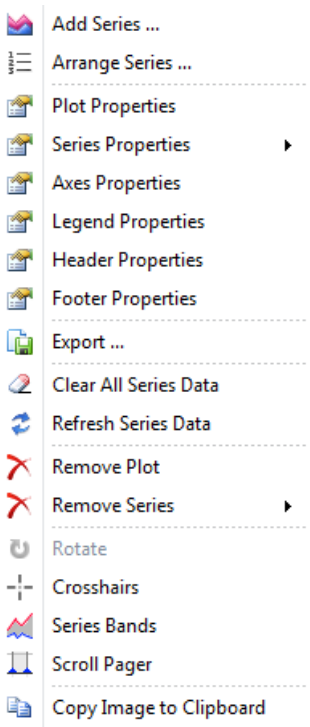


Figure 2.68 Context menu for a Plot window

Properties of the Axes

The axes properties can be accessed through the Properties window, by selecting one of the axes on the Plot; or through the Context Menu (cf. Figure 2.68) by choosing the corresponding item.

The main Properties of an axis (Bottom, Depth, Left, Right or Top) are:

- **Automatic**:
  the scale is set automatically (True) or not (False)
- **AutomaticMaximum**:
  the maximum value of the scale is set automatically (True) or not (False)
- **AutomaticMinimum**:
  the minimum value of the scale is set automatically (True) or not (False)
- **GridVisible**:
  the grid lines are visible (True) or not (False)
- **Labels**:
  the axis labels and their sub-properties
- **Logarithmic**:
  the scale is logarithmic (True) or not (False)

- **Maximum**:
  maximum value of the scale (cf. AutomaticMaximum)
- **Minimum**:
  minimum value of the scale (cf. AutomaticMinimum)
- **Title**:
  title of the axis; the usual properties of a text object are available as sub-properties

Properties of the Legend

The legend properties can be accessed through the Properties window, by selecting the Legend on the Plot; or through the Context Menu (cf. Figure 2.68) by choosing the corresponding item.

The main Properties of the legend are:

- **Alignment**:
  position of the Legend in relation to the Chart area
- **Checkboxes**:
  whether check boxes are to be displayed next to the symbol of each data series
- **Visible**:
  whether the Legend is to be displayed or not

Properties of a Data Series

The properties of a Data Series can be accessed through the Properties window, by selecting the specific series on the Plot; or through the Context Menu (cf. Figure 2.68) by choosing the corresponding item.

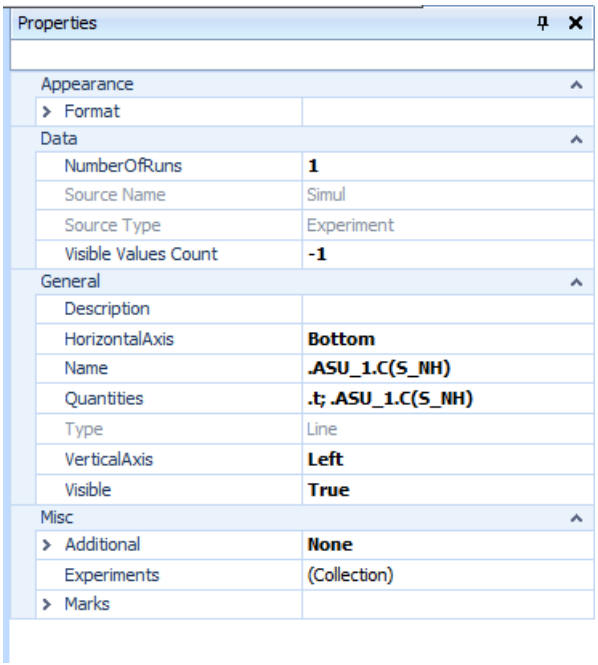


Figure 2.69 Properties of a Data Series of a Plot

Note that the list shown in Figure 2.69 and illustrated hereafter is specific to a Line type Data Series in a Trajectory type Plot: other Plot types may have different properties.

The main Properties of a Data Series are (cf. Figure 2.69):

- **Format**:
  groups several properties related to the appearance of the data series, such as Color, LineStyle, LineWidth, PointStyle etc.
- **NumberOfRuns**:
  clears the data after the specified number of runs.
- **Source Name**:
  name of the data source, e.g. ‘Simul’ (for the simulation experiment) or the path to the file containing the data (read-only).
- **Source Type**:
  type of the data source, e.g. ‘Experiment’ or ‘File’ (read-only).
- **Visible Values Count**:
  number of data points (starting from the last one) to be shown on the plot. Works for Time series: “-1” means all points; “1” means only the last one.
- **Description**:
  free text field in which an informative description of the quantity can be inputted.

- **HorizontalAxis**:
  position of the x-axis, i.e. Bottom (default) or Top.
- **Name**:
  name assigned to the Data Series; by default, the variable name, but can be renamed for convenience.
- **Quantities**:
  the model variable that is represented; it is assigned by dragging the variable from the Block Details pane onto the Plot; or by selecting the Data Series in the Add Data Series dialog window (cf. Section 5.7.3).
- **Type**:
  how the Data Series is represented on the Plot, e.g. Line; it depends on the type of Plot (read-only).
- **VerticalAxis**:
  position of the y-axis, i.e. Left (default) or Right.
- **Visible**:
  attribute to turn on (True, default) or off (False) the data series.
- **Marks**:
  groups several properties that apply to the markers for the individual data points.

2.10.15 Properties of a Table Pane

A Table pane acts as a *container* (or sub-container, since the Sheet is the overall container for Dashboard widgets) on which several variables can be dragged and will result in a data column.

The main Properties of a Table pane are:

- **Name**:
  name of the Table pane (as the container).
- **ShowCaption**:
  whether the caption is to be displayed above the pane.

A Context Menu is displayed by clicking on blue arrow in the upper right corner of the Table pane which gives access to the following items (Figure 2.71):

- **Add Columns**:
  allows for adding a column to the table by selecting the corresponding item from the treeview shown in Figure 2.70. This is alternative to dragging a model parameter or variable from the Block Details pane. It is the only way to add a column, in case of quantities that are computed (and created) at runtime (e.g. Time series Criteria).

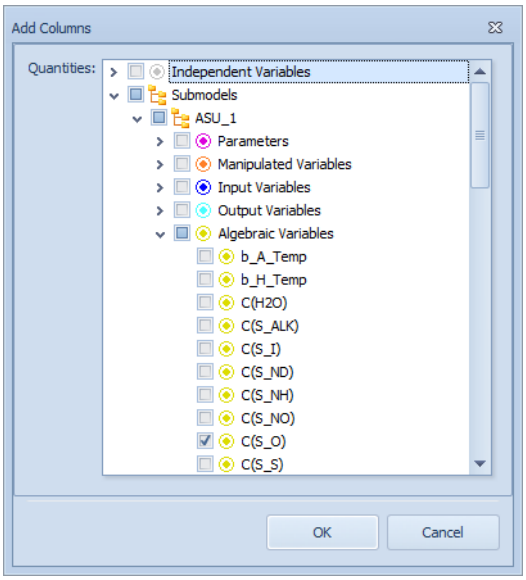


Figure 2.70 Add Columns dialogue for a Table pane

- **Table Properties**:
  shows the properties of the Table Pane, i.e. its Name.
- **Column Properties**:
  further visualises a sub-menu containing the list of the available Columns.

  The Column properties are:

  - Description:
    a free text field in which an informative description of the quantity can be inputted.
  - Format Specifier:
    the number format specifier
  - Name:
    name that is assigned to the column; by default, the variable name, but can be renamed for convenience.
  - Quantities:
    the model variable that is represented; it is assigned by dragging the variable from the Block Details pane onto the Table.
  - Unit:
    (read-only) default unit for the variable, as it is specified in the model library.
  - Visible:
    attribute to make the column visible (True, default) or invisible (False).

- **Export**:
  exports the content of the Table to various formats.

- **Print**:
  prints the content of the Table.
- **Clear All Data Series**:
  clears all data series.
- **Refresh Data Series**:
  refreshes all data series.
- **Remove Table**:
  removes the Table pane.
- **Remove Columns**:
  further visualises a sub-menu containing the list of the Columns to be individually removed.
- **Copy Values to Experiment**:
  copies the selected values to the experiment.
- **Copy Table to Clipboard**:
  copies the table to the clipboard

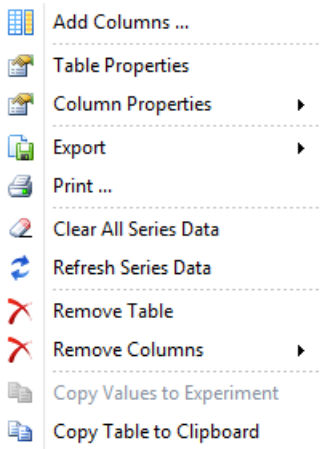


Figure 2.71 Context menu for a Table pane

2.11 Shortcuts

A number of key actions that are likely to be often executed in the process of setting up a model and running dynamic simulations have been associated to keyboard shortcuts.

The following tables present an overview of the shortcuts available in WEST to open one of the System Windows (Table 2.4) and to open/close projects, start/stop simulations, etc. (Table 2.5).

Table 2.4 List of shortcuts for the System Windows in WEST

System window	Shortcut
Block Details	Ctrl+W, D
Block Library	Ctrl+W, X
Block Summary	Ctrl+W, S
Control Center	Ctrl+W, C
Getting Started	Ctrl+W, G
Layers	Ctrl+W, Y
Layout	Ctrl+W, A
Layout Overview	Ctrl+W, O
Logging	Ctrl+W, L
Model Explorer	Ctrl+W, E
Properties	Ctrl+W, P
Analysis Properties	Ctrl+W, Q
Simulation Properties	Ctrl+W, Z
Notes	Ctrl+W, N

Table 2.5 Action shortcuts in WEST

Action	Shortcut
Close Application (Exit)	Alt+F4
Help	F1
New Project	Ctrl+N
Open Project	Ctrl+O
Save Project	Ctrl+S
Close Project	Ctrl+F4
Simulation Start/Pause	F6
Simulation Stop	F7
Full Screen	Shift+Alt+Enter

WEST - © DHI A/S

3 The WEST Model

A mathematical model in WEST, i.e. the model underlying the Layout, consists of three parts: the Input Model, the Plant Model and the Output Model (cf. Figure 3.1). The input to the overall model is the form of an input file; optionally, an output file can be present too.

The Input and Output Models are to be implemented by executing the Influent and Effluent Tool respectively (cf. Section 2.9), whereas the Plant Model is generated by setting up the Plant Layout.

Unlike any other model of a Block in WEST, the input and output models are not part of the model library. Instead they are generated at the Project level and can be found in the Project folder.

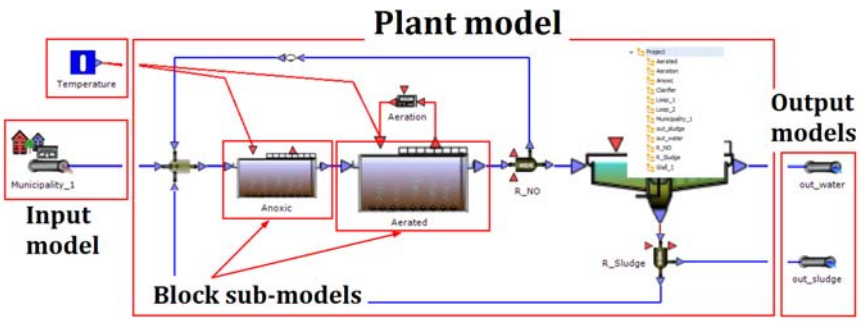


Figure 3.1 The elements of a WEST Model

3.1 The Input Model

The input to a WEST model is a **component vector** containing the model components (variables) that are specific to the given Model Category, e.g. ASM1.

Furthermore, all components are to be expressed as fluxes, with unit g/d.

In practice, however, data are generally available in more convenient quantities, e.g. COD, TSS, etc.; and measurements are expressed as flow rates (m^3/d) and concentrations (mg/l). Therefore, every Input Model (i.e. the model behind an Input Block) consists of the following elements:

- **a Fractionation Model**:
  an interface describing how the measured quantities are mapped to the Model Components

- **a Conversion Model**:
  which translates the flow rate and concentration units into flux (load) units
- **two input files**:
  a dynamic time series that is generated by combining individual data sources (as described in 'Step 3: Data Import' in Section 2.9.5) and the corresponding steady-state time series.

If the model components however are used directly for the definition of the component vector and/or the data series are provided in flux units, the Fractionation Model and/or the Conversion Model are left out.

3.2 The Plant Model

The Plant Layout, i.e. the graphical representation of the plant on the Layout Sheet, is automatically translated by WEST in the corresponding Plant Model by combining the individual sub-models associated to each Block.

3.2.1 Flattened Model

Once the Plant Layout is complete, and its Model generated and ready for the simulation, the model compiler creates an executable model which ensures optimal simulation speed.

The following steps are carried out to generate the executable model:

- Translation of the Plant Layout to the Model in MSL
- Flattening and optimisation of the MSL model
- Translation of the flat optimised MSL Model to C
- Compilation and linking of the C code

3.2.2 Parameters

In the WEST Model Library, a *functional* distinction is made between Parameters and Manipulated Variables (Table 3.1).

Both quantities denote Model Parameters (with respect to the usual description of models in literature) and as such, they are both displayed in the Parameters tab of the Block Details window.

However, Manipulated Variables can be altered during the simulation and therefore are in essence input variables (interface variables, in the MSL language); whereas Parameters can only be altered prior to starting a simulation (their value is constant during the simulation).

As Model Parameters, both quantities can be attached to an Input Field or a Slider (cf. Section 2.6.7), but for a Parameters these will be disabled during the simulation. Only Manipulated Variables can receive input data (through a Data Input) or be controlled (through a Controller or a Timer) – and will therefore be exposed in the corresponding Interface Link dialog window.

Table 3.1 Ways to alter model Parameters and Manipulated Variables

	'Offline' = prior to simulation	**'Online' = during simulation**
Parameters	Yes	No
Manipulated Variables	Yes	Yes
	Slider Input Field Radiobutton Combobox Checkbox	Slider Input Field Radiobutton Combobox Checkbox Controller Block Timer Block Input File

3.2.3 Top-Level Parameters

It is often the case that several sub-models within the overall Plant Model have a common (sub)set of Parameters, typically (but not exclusively) the kinetic and stoichiometric parameters: for instance, a series of CSTR units.

If these parameters were to be assigned the same values across the different units (which is generally the case), it is very inconvenient to assign or change their values simultaneously in every Block.

WEST allows for grouping the parameters that are common to different Blocks into a set of the corresponding parameters at the level of the overall Plant Model (as opposed to the Block level). The new set of parameters is therefore termed Top-Level Parameters and will be accessible through the Block Details window of the Project (cf. Figure 3.2).

A value assigned to a Top-level Parameter is automatically transferred to the sub-model parameters that are linked to that Parameter.

See also Section 5.2.12.

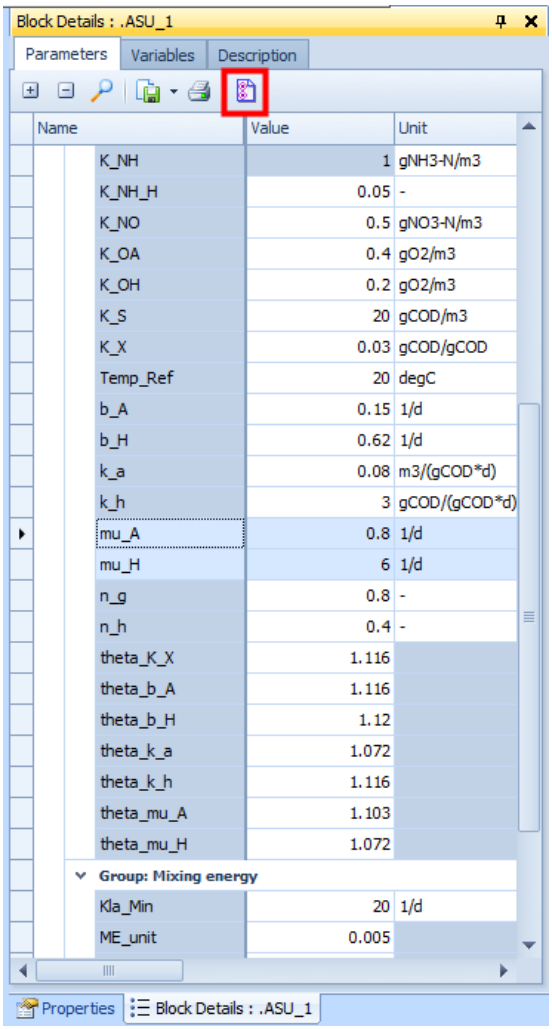


Figure 3.2 Parameters tab page of the Plant Model, showing the top-level parameters

3.2.4 Top-level Interface Variables

An explicit handle at the level of the overall model (hence: top-level interface variable) is required, for instance, to pass a scalar data input, or receive a scalar data output, to any of the sub-models that compose the plant model.

More generally, a top-level interface variable exposes an interface variable of a sub-model, making it usable to the outside world: the overall model then becomes a fully qualified model that could potentially be used as “composite” sub-model (coupled model) within a larger model.

To define a top-level interface variable, choose the corresponding item in the Context menu (Figure 3.3) and fill in the following properties (Figure 3.4).

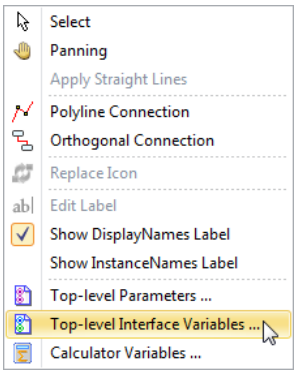


Figure 3.3 Context menu to activate the dialog to create a Top-level Interface Variable

- **Name**
  Name of the variable.
- **Causality**
  Causality: “Input” (e.g. if it is to be associated to a Data Input Block) or “Output” (e.g. if it is to be associated to a Data Output Block).
- **Type**
  Data type: not mandatory (default is “Real”) but useful to allow for automatic unit conversion.
- **Description** (optional)
- **Unit**
  Not mandatory (default is unitless) but useful to allow for automatic unit conversion.
- **Default Value**
  Not mandatory (default is 0) but useful to allow for automatic unit conversion.
- **Group**
  Not mandatory but necessary to have variables grouped in the Block Details pane.
- **Terminal**
  Not mandatory but necessary to expose the variable through a terminal, if the plant model is used as coupled model.
- Lower Bound
- Upper Bound
- **Manipulated**
  Indicates whether the variable can be manipulated in the scope of a coupled model.
- **Favorite**
  Indicates whether the variable is to be displayed in the Block Summary pane, in the scope of a coupled model.

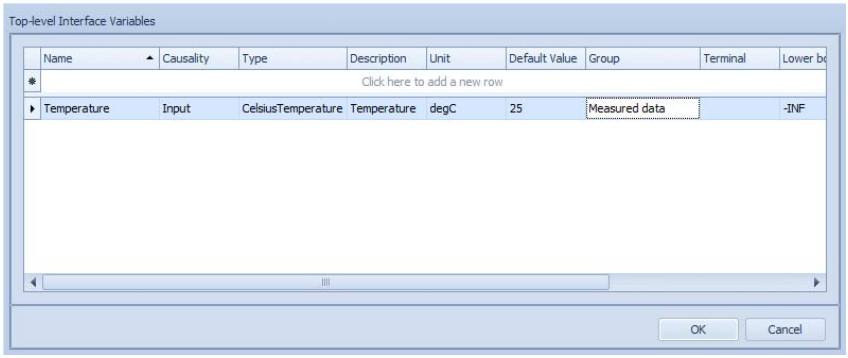


Figure 3.4 Top-level Interface Variables dialog window

3.2.5 Calculator Variables

Variables that are a combination of other model Variables (hence termed: Calculator Variables) can be added to the Project and will be treated as any other Model Variable.

Since a Calculator Variable can be defined as a combination of any Variable of the model, i.e. variables belonging to different sub-models can be assembled into one composite variable, a Calculator Variable belongs to the overall model and will therefore be accessible through the Block Detail window of the Project (cf. Figure 3.5).

Calculator Variables may be useful for instance to evaluate performance indices which typically are defined by taking into account several (concentration) variables in the effluent.

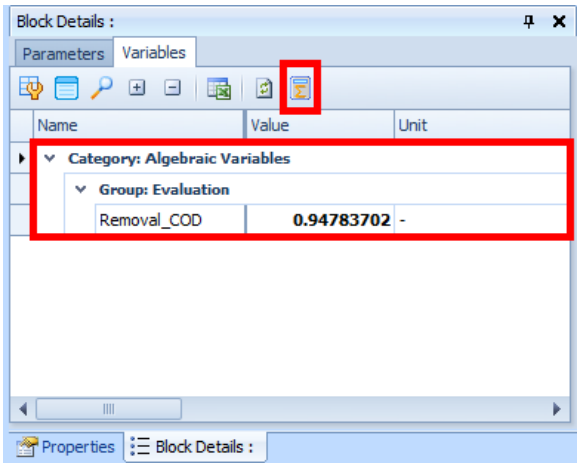


Figure 3.5 Variables tab page of the Plant Model, showing a Calculator Variable

A Calculator Variable can be implemented by choosing the corresponding item in the Context menu and fill in the following properties (Figure 3.6):

- **Name**
  Name of the variable.
- **Expression**
  Insertion of quantity names is facilitated through a dropdown that is activated by typing “.”
- **Description**
  (optional) description of the Variable
- **Unit**
  Not mandatory (default is unitless) but useful to allow for automatic unit conversion.
- **Default Value**
  Not mandatory (default is 0) but useful to allow for automatic unit conversion.
- **Group**
  Not mandatory but necessary to have variables grouped in the Block Details pane.
- Lower Bound
- Upper Bound

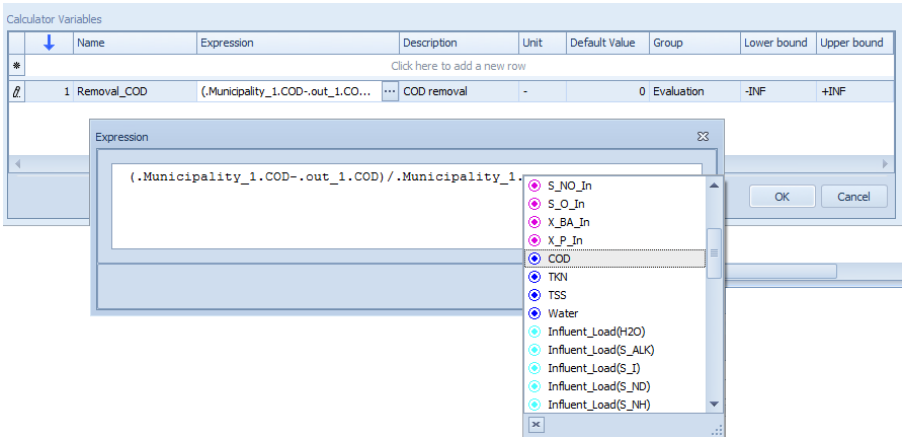


Figure 3.6 Calculator Variable Expression editor

3.3 The Output Model

A WEST Plant Model generates output data, in the same format as the input data that are required (cf. Section 3.2 ‘The Plant Model’), i.e. specified as model components (i.e. ASM variables) and expressed as fluxes (g/d).

In practice, however, effluent data are generally to be compared against legislation standards to evaluate the performance of the plant: it is therefore more convenient that they be expressed in terms of concentrations (mg/l) of COD, BOD, TSS, TN, etc. Therefore, every Output Model (i.e. the model behind an Output Block) consists of the following elements:

- **a De-fractionation Model**:
  an interface describing how the Model Components are combined into the measured quantities
- **a Conversion Model**:
  which translates the flux (load) units into flow rate and concentration units

If the model components, however, are to be used directly for the definition of the output vector, the De-Fractionation Model is left out.

4 Experiment Types

This chapter discusses the types of advanced experiments that are available in WEST. The term advanced (or virtual) experiment is used here to refer to a computational procedure that is performed on the executable version of a (plant) model. The most straightforward virtual experiment type is simulation, but several other, more advanced procedures are available in WEST.

When a new project is created, two separate simulation experiments are automatically created behind the scenes. The first is intended for simulating the model under steady-state conditions, while the other for performing dynamic simulations. Both experiments share the same model, but can be configured separately in terms of parameter values, initial conditions, solver settings, etc. as will be discussed later in this chapter. The steady-state and dynamic experiments can also automatically compute a set of predefined criteria on a selected number of time series. Also these time series criteria can be configured separately for both simulation experiments.

A project can be upgraded from its original status to include one of the following advanced computational procedures:

- **Local Sensitivity Analysis Experiment (LSA)**:
  Uses the finite difference approach to study the effect of small changes in parameters to output variables.
- **Global Sensitivity Analysis Experiment (GSA)**:
  Runs a series of simulation experiments using parameter vectors sampled through Latin Hypercube Sampling, and performs Linear Regression.
- **Parameter Estimation Experiment (PE)**:
  Minimises an objective by running simulations using different values for a selected set of parameters, typically to match simulated to measured data.
- **Scenario Analysis Experiment (SA)**:
  Runs a series of simulation experiments using different values for a selected set of parameters.
- **Uncertainty Analysis Experiment (UA)**:
  Runs a series of simulation experiments using parameter vectors sampled through Latin Hypercube Sampling, and computes various types of aggregated data.

When a project is upgraded to support one of the above virtual experiment types, a choice has to be made whether to base the upgraded project on the original steady-state or the original dynamic simulation experiment. As of that moment, the settings of the selected simulation experiment are in effect in the newly created virtual experiment. A project can also be downgraded back to the original situation with the steady-state and dynamic simulation experi-

ment, not including any of the advanced computational procedures listed above.

4.1 Steady-State and Dynamic Experiment

For any layout composed in WEST, two simulation experiments will be automatically created. One is named Steady-state and is intended for studying the system under steady-state conditions. The other experiment is named Dynamic and is intended for studying the system under dynamic conditions. These experiments use separate influent files, which are generated by the Interface Tools. Also, the values of derived variables obtained at the end of the steady-state simulation, will be used as initial conditions to the dynamic simulation (although this behaviour can be optionally altered by the user if needed).

4.1.1 Simulation Properties

The steady-state and dynamic simulation experiments have their own separate set of properties, which can be accessed through the Simulation button in the Properties group of the Project menu, which will cause the dialog window shown in Figure 4.1 to be displayed.

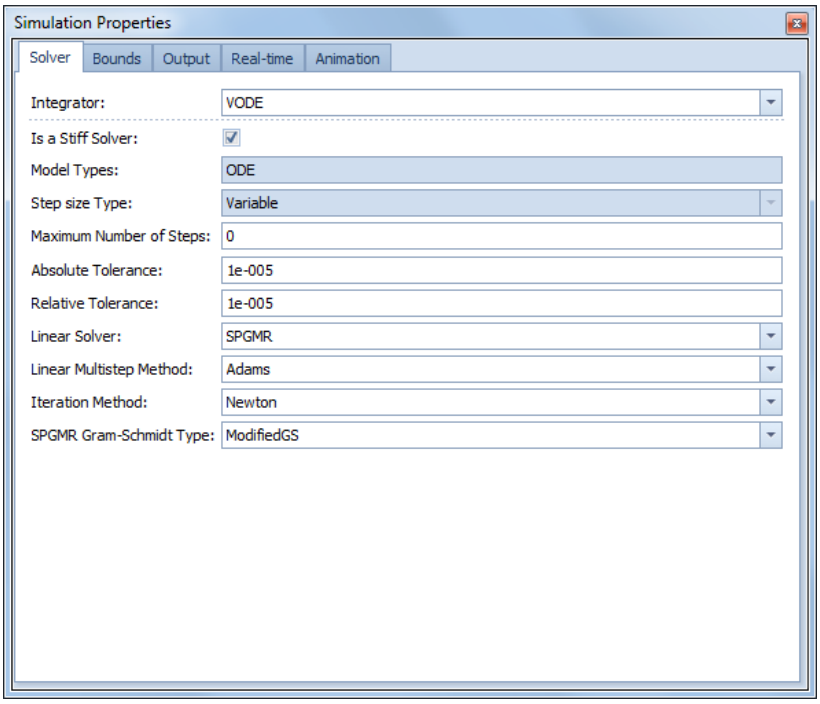


Figure 4.1 The Solver tab page of the Simulation Properties window

Solver Tab Page

The Solver tab page allows for selecting an integration method and for setting its properties. The following integration methods are available:

- **Alg**:
  Algebraic solver, only to be used for systems without differential equations
  - *Model Types*: Read-only, informational property
  - *Step size Type*: Read-only, informational property
  - *Maximum Number of Steps*: Maximum number of steps the integrator can take
  - *Step size*: Size of the steps the integrator should take

- **VODE**:
  Advanced stiff solver
  - *Is a Stiff Solver*: Read-only, informational property
  - *Model Types*: Read-only, informational property
  - *Step size Type*: Read-only, informational property
  - *Maximum Number of Steps*: Maximum number of steps the integrator can take
  - *Absolute Tolerance*: Absolute tolerance
  - *Relative Tolerance*: Relative tolerance
  - *Linear Solver*: Linear solver method. The options are:
    a. Band
    b. Dense
    c. Diag
    d. SPGMR (Scaled Preconditioned Generalised Minimal Residual Method)
  - *Linear Multistep Method*: Linear multistep method. The options are:
    a. Adams
    b. DBF
  - *Iteration Method*: Iteration method. The options are:
    a. Functional
    b. Newton
  - *SPGMR Gram-Schmidt Type*: SPGMR Gram-Schmidt type. Only relevant if the linear solver is set to SPGMR
  - *Lower Bandwidth*: Lower bandwidth. Only relevant if the linear solver is set to Band.
  - *Upper Bandwidth*: Upper bandwidth. Only relevant if the linear solver is set to Band.

- **RK4**:
  Runge-Kutta 4
  - *Is a Stiff Solver*: Read-only, informational property
  - *Model Types*: Read-only, informational property
  - *Step size Type*: Read-only, informational property

  - *Maximum Number of Steps*: Maximum number of steps the integrator can take
  - *Step size*: Size of the steps the integrator should take

- **RK4ASC**:
  Runge-Kutta 4 with Adaptive Step size Control
  - *Is a Stiff Solver*: Read-only, informational property
  - *Model Types*: Read-only, informational property
  - *Step size Type*: Read-only, informational property
  - *Maximum Number of Steps*: Maximum number of steps the integrator can take
  - *Accuracy*: Accuracy
  - *Check Minimum Step size*: Determines whether the step size is bound by the Minimum Step size setting
  - *Initial Step size*: Initial step size
  - *Minimum Step size*: Minimum step size
  - *Maximum Step size*: Maximum step size

Bounds Tab Page

The Bounds tab page has 2 checkbuttons:

- **Check Bounds**:
  Indicates whether current values of parameters and variables have to be checked during the simulation process against lower and upper bounds that have been set as part of the model description. This flag is on by default. Switching it off will favour simulation performance, but may make a user unaware of important bound exceedances.

- **Stop When Bounds Violation**:
  Indicates whether the simulation process should be halted when a bound violation is detected. This flag is off by default.

Output Tab Page

The Output tab page allows for configuring the simulation output channels. Available channel types are the following:

- **File**:
  One or more simulation output files can be configured. Each file has its own set of properties:
  - *Enabled*: Activates file output
  - *File name*: Name of the output file
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic

- Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
- Number of Communication points: Number of output time points
- *Communication Interval*: Interval between output time points
- *Interpolated*: Data interpolation on or off
- *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)
- *Quantities*: Model quantities that are to be sent to output. These can either be added through the + and – buttons, or through drag-and-drop actions from the Block Details or Block Summary Windows.

- **Buffer**:
  Internal simulation output buffer (only relevant in the scope of WEST-forAUTOMATION). Properties to be specified are the following:
  - *Enabled*: Activates buffer output
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic
  - Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
  - Number of Communication points: Number of output time points
  - *Communication Interval*: Interval between output time points
  - *Interpolated*: Data interpolation on or off
  - *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)
  - *Quantities*: Model quantities that are to be sent to output. These can either be added through the + and – buttons, or through drag-and-drop actions from the Block Details or Block Summary Windows.

- **Field**:
  Simulation output fields (cf. Dashboard). Properties to be specified are the following:
  - *Enabled*: Activates field output
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic
  - Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
  - Number of Communication points: Number of output time points
  - *Communication Interval*: Interval between output time points
  - *Interpolated*: Data interpolation on or off

- *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)

- **Plot**:
  Simulation output plots (cf. Dashboard). Properties to be specified are the following:
  - *Enabled*: Activates plot output
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic
  - Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
  - Number of Communication points: Number of output time points
  - *Communication Interval*: Interval between output time points
  - *Interpolated*: Data interpolation on or off
  - *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)

- **Table**:
  Simulation output tables (cf. Dashboard). Properties to be specified are the following:
  - *Enabled*: Activates table output
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic
  - Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
  - Number of Communication points: Number of output time points
  - *Communication Interval*: Interval between output time points
  - *Interpolated*: Data interpolation on or off
  - *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)

- **Gauge**:
  Simulation output gauges (cf. Dashboard). Properties to be specified are the following:
  - *Enabled*: Activates gauge output
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic

- Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
- Number of Communication points: Number of output time points
- *Communication Interval*: Interval between output time points
- *Interpolated*: Data interpolation on or off
- *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)

- **Animation**:
  Layout simulation output. Properties to be specified are the following:
  - *Enabled*: Activates animation output
  - *Start time*: Start of the time window
  - *Stop Time*: End of the time window
  - *Communication Interval Type*:
    a. Linear
    b. Logarithmic
  - Specify Number of Communication Points: Indicates whether a number of communication points rather than a communication interval will be specified
  - Number of Communication points: Number of output time points
  - *Communication Interval*: Interval between output time points
  - *Interpolated*: Data interpolation on or off
  - *Use Display Units*: Switches between System Units (default units of the model library) and Display Units (selected in the GUI)

Real-Time Tab Page

The Real-time tab page allows for synchronizing the elapsed simulated time with elapsed simulation (i.e. real) time. The page has 1 checkbutton and 1 entry field:

- **Real-time Simulation**:
  Indicates whether elapsed simulated time should be synchronized with elapsed real time. This flag is off by default. For typical water quality projects, switching on this flag will cause the simulation to proceed extremely slowly, due to the high discrepancy between simulated (typically days, weeks or years) and real time (typically seconds to minutes).

- **Speed-up Factor**:
  A value higher than 1 will speed up the real-time simulation, still keeping the elapsed simulated time proportional to the elapsed real time. For typical applications, values in the 10,000 – 1,000,000 range are often used here.

Animation Tab Page

The Animation tab page allows for configuring a number of dynamic (simulation-controlled) animations in relation to the layout. Each animation type can be configured through a separate sub-tab page:

- **Line Thickness**:
  Allows for associating the line thickness of layout connections with the value of a component of the category's component vector. The following settings are available:

  - *Enabled*: Indicates whether line thickness animation is to be activated.
  - *Component Name*: Combobox that allows for selecting the component that is to be associated with line thickness.
  - *Conversion Type*: Combobox that allows for selecting the conversion – if any - that needs to be applied to the selected component's values prior to their further use. Options are "None" (no conversion), "F2C" (flux to concentration) or "C2F" (concentration to flux).
  - *Lookup Table*: Indicates whether a lookup table rather than a function is to be used to map (converted) component values to line thicknesses. If this flag is switched on, a table appears with columns labeled as "Value" and "Thickness". By adding rows to this table, the desired mapping can be defined. If a lookup table is in effect, the other line thickness settings listed hereunder are irrelevant and will hence not be shown.
  - *Type*: Type of the mapping function. Options are "Linear" or "Logarithmic".
  - *Inverse*: Indicates whether the mapping function is to be inverted (highest component value associated with smallest line thickness and vice versa).
  - *Visual Minimum*: Smallest line thickness that is to be adopted.
  - *Visual Maximum*: Largest line thickness that is to be adopted.
  - *Automatic Real Minimum*: Indicates whether, at every simulation time point, the smallest component value (after conversion) across the entire layout is to be determined and hence associated with the Visual Minimum.
  - *Real Minimum*: User-defined component value that is to be associated with the Visual Minimum.
  - *Automatic Real Maximum*: Indicates whether, at every simulation time point, the largest component value (after conversion) across the entire layout is to be determined and hence associated with the Visual Maximum.
  - *Real Maximum*: User-defined component value that is to be associated with the Visual Maximum.

- **Line Color**:
  Allows for associating the line color of layout connections with the value of a component of the category's component vector. The following settings are available:

  - *Enabled*: Indicates whether line color animation is to be activated.
  - *Component Name*: Combobox that allows for selecting the component that is to be associated with line color.
  - *Conversion Type*: Combobox that allows for selecting the conversion – if any - that needs to be applied to the selected component's values prior to their further use. Options are "None" (no conversion), "F2C" (flux to concentration) or "C2F" (concentration to flux).
  - *Lookup Table*: Indicates whether a lookup table rather than a function is to be used to map (converted) component values to line colors. If this flag is switched on, a table appears with columns labeled as "Value" and "Color". By adding rows to this table, the desired mapping can be defined. If a lookup table is in effect, the other line color settings listed hereunder are irrelevant and will hence not be shown.
  - *Color Parameter*: Aspect of the color (according to the HSL color model) that is to be associated with (converted) component values. Options are "Hue", "Saturation" and "Luminosity".
  - *Color*: Reference color to which modifications of either hue, saturation or luminosity (see Color Parameter) are to be applied.
  - *Type*: Type of the mapping function. Options are "Linear" or "Logarithmic".
  - *Inverse*: Indicates whether the mapping function is to be inverted (highest component value associated with smallest line color parameter value and vice versa).
  - *Visual Minimum*: Smallest line color parameter value that is to be adopted.
  - *Visual Maximum*: Largest line color parameter value that is to be adopted.
  - *Automatic Real Minimum*: Indicates whether, at every simulation time point, the smallest component value (after conversion) across the entire layout is to be determined and hence associated with the Visual Minimum.
  - *Real Minimum*: User-defined component value that is to be associated with the Visual Minimum.
  - *Automatic Real Maximum*: Indicates whether, at every simulation time point, the largest component value (after conversion) across the entire layout is to be determined and hence associated with the Visual Maximum.
  - *Real Maximum*: User-defined component value that is to be associated with the Visual Maximum.

- **Labels**:
  Allows for displaying the value of one or more components of the category's component vector on layout connections. The following settings are available:

  – *Enabled*: Indicates whether labels are to be activated.
  – *Components*: List of components for which values have to be displayed in the label.
  – *Conversion Type*: Combobox that allows for selecting the conversion – if any - that needs to be applied to the selected component's values prior to their further use. Options are "None" (no conversion), "F2C" (flux to concentration) or "C2F" (concentration to flux).
  – *Show Component Names*: Indicates whether the label should contain values prefixed with the name of the component.
  – *Format Specifier*: Format specifier to be applied values as they are shown in the label.
  – *Background Color*: Label background color.
  – *Font Name*: Label font name.
  – *Font Size*: Label font size.
  – *Font Color*: Label text color.
  – *Line Color*: Label line color.
  – *Line Size*: Label line thickness.

- **Alerts**: Allows for highlighting layout blocks (on the basis of a red bounding box) in which one or more user-specified bounds are violated. The following settings are available:

  – *Enabled*: Indicates whether alerts are to be activated.
  – *Patterns*: List of parameter / variable name patterns and associated lower and upper bounds that are to be checked. Patterns are to be specified as regular expressions according to the ECMA standard (see http://www.ecma-international.org/publications/standards/Ecma-262.htm). When a violation is detected, a red bounding box is draw around the block in which it occurs. Further information on the exact nature of the violation (time, parameter / variable name and details of the exceedance) can be found in the Alerts tab of the Block Summary Window.

Note that for any type of animation to work, a necessary condition is that values are sent to the animation engine through the Animation Simulation Output Channel (cf. Output tab). This channel therefore needs to be enabled. As animation can have a detrimental effect on simulation performance, it is important to choose the communication interval (or number of communication points) of the Animation channel wisely.

4.1.2 Analysis Properties

The steady-state and dynamic simulation experiments also allow for computing a number of criteria on the basis of the simulated time series. These crite-

ria can be configured in the Analysis Properties dialogue, accessible through the Project | Properties Menu.

The tab pages of the Analysis Properties window for the selected Simulation Experiment are described below.

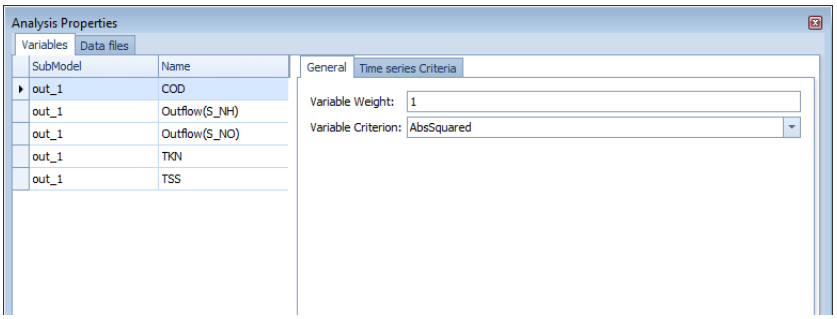


Figure 4.2 The General tab page of the Analysis Properties dialogue (Dynamic Simulation experiment)

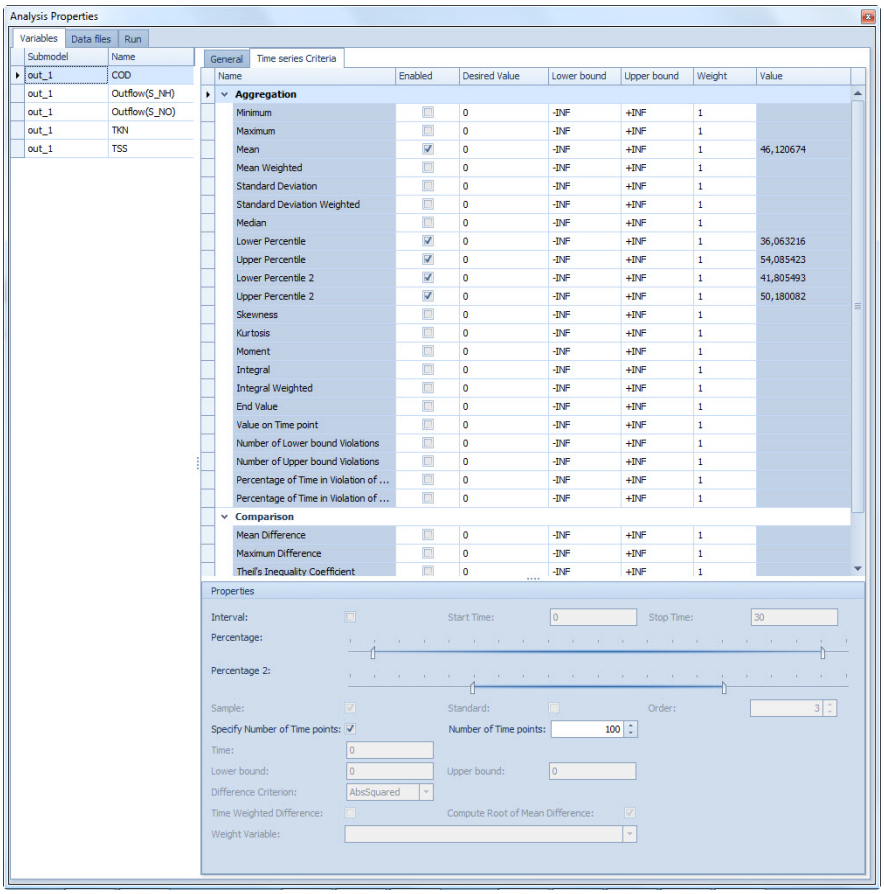


Figure 4.3 The Time series Criteria tab page of the Analysis Properties dialogue (Dynamic Simulation experiment)

Variables

- **Left pane** (Figure 4.2):
  List of variables for which criteria are to be computed. Items can be added to this list through drag-and-drop from the Block Details- and the Block Summary windows, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  Settings for the variable selected in the left pane.

  – ***General*** (Figure 4.2):
    a. Variable Weight: Weight that is applied to the aggregation of the variable's sub-criteria in order to compute the overall aggregated objective value.

    b. Variable Criterion: Method that is to be used to compute the difference between the variable's sub-criteria and their desired values.

The options are:
AbsSquared: Squared difference
RelSquared: Squared relative difference
Abs: Absolute difference
Rel: Relative difference

– ***Time series Criteria*** (Figure 4.3)
An overview of the Time series Criteria is presented in Table 4.1 and Table 4.2.

Table 4.1 Time series Criteria: Aggregation

Criterion and Properties	Description
Minimum	Minimum of the time series.
Maximum	Maximum of the time series.
Mean	Time-weighted mean of the time series.
Mean Weighted	Similar to the time-weighted mean criterion. However in this case also the Weight Variable is taken into account as a weight.
Weight Variable	Weight, typically: the flow of the solvent.
Standard Deviation	Time-weighted standard deviation of the time series.
Standard Deviation Weighted	Standard deviation of the time series, computed through weighing according to time and another variable that is specified as Weight Variable.
Weight Variable	Weight, typically: the flow of the solvent.
Median	Median of the time series.
Specify Number of Time Points	Switches between Number of Time points and Time Interval.
Number of Time points	Number of time points.
Time interval	Time interval.
Lower Percentile	Lower percentile of the time series.
Percentage	Lower percentage.
Specify Number of Time Points	Switches between Number of Time points and Time Interval.
Number of Time points	Number of time points.
Time interval	Time interval.
Upper Percentile	Upper percentile of the time series.
Percentage	Upper percentage.
Specify Number of Time Points	Switches between Number of Time points and Time Interval.
Number of Time points	Number of time points.
Time Interval	Time interval.

Table 4.1 Time series Criteria: Aggregation

Criterion and Properties	Description
Lower Percentile 2	Second lower percentile, to be configured in a way similar to the first lower percentile.
Percentage 2	Second lower percentage.
Specify Number of Time Points	Switches between Number of Time points and Time Interval.
Number of Time points	Number of time points.
Time Interval	Time interval.
Upper Percentile 2	Second upper percentile, to be configured in a way similar to the first upper percentile
Percentage 2	Second upper percentage
Specify Number of Time Points	Switches between Number of Time points and Time Interval
Number of Time points	Number of time points
Time Interval	Time interval
Skewness	Skewness of the time series
Sample	Switches between Sample and Population skew-ness
Specify Number of Time Points	Switches between Number of Time points and Time Interval
Number of Time points	Number of time points
Time Interval	Time interval
Kurtosis	Kurtosis of the time series.
Sample	Switches between Sample and Population kurtosis.
Specify Number of Time Points	Switches between Number of Time points and Time Interval.
Number of Time points	Number of time points.
Time Interval	Time interval.
Moment	n-th order moment of the time series.
Standard	Switches between Standard and non-Standard moment.
Order	Order of the moment.
Specify Number of Time Points	Switches between Number of Time points and Time Interval.
Number of Time points	Number of time points.
Time Interval	Time interval.
Integral	Time-weighted integral of the time series.
Integral Weighted	Integral of the time series, computed through weighing according to time and another variable that is specified as Weight Variable.

Table 4.1 Time series Criteria: Aggregation

Criterion and Properties	Description
Weight Variable	Weight, typically: the flow of the solvent.
End Value	End value of the time series.
Value on Time point	Value corresponding with a particular time point of the time series.
Time	Time point.
Number of Lower bound Violations	Number of times the time series violates a particular lower bound.
Lower bound	Lower bound.
Number of Upper bound Violations	Number of times the time series violates a particular upper bound.
Upper bound	Upper bound.
Percentage of Time in Violation of Lower Bound	Percentage of time that the time series is in violation of a particular lower bound.
Lower bound	Lower bound.
Percentage of Time in Violation of Upper Bound	Percentage of time that the time series is in violation of a particular upper bound.
Upper bound	Upper bound.

Table 4.2 Time series Criteria: Comparison

Criterion and Properties	Description
Mean Difference	Mean difference between one or more reference time series and the simulated time series. The reference time series are to be specified as data files.
Difference Criterion	Criterion used during residue computations. The options are: - AbsSquared: Squared difference - RelSquared: Squared relative difference - Abs: Absolute difference - Rel: Relative difference
Time Weighted Difference	Switches between time weighted and non-time weighted differences. Time weights are to be specified in the reference time series data files.
Compute Root of Mean Difference	Indicates whether the root of the mean difference is to be computed.
Maximum Difference	Maximum difference between one or more reference time series and the simulated time series. The reference time series are to be specified as data files.

Table 4.2 Time series Criteria: Comparison

Criterion and Properties	Description
Difference Criterion	Criterion used during residue computations. The options are: -AbsSquared: Squared difference - RelSquared: Squared relative difference - Abs: Absolute difference - Rel: Relative difference
Theil's Inequality Coefficient (TIC)	This is another measure for the difference between one or more reference time series and the simulated time series. The reference time series are to be specified as data files. The most interesting feature of TIC is that it always yields a value between 0 and 1.
End Value Difference	Difference between the end value of the reference- and of the simulated time series. The reference time series is to be specified as data files.

Data Files

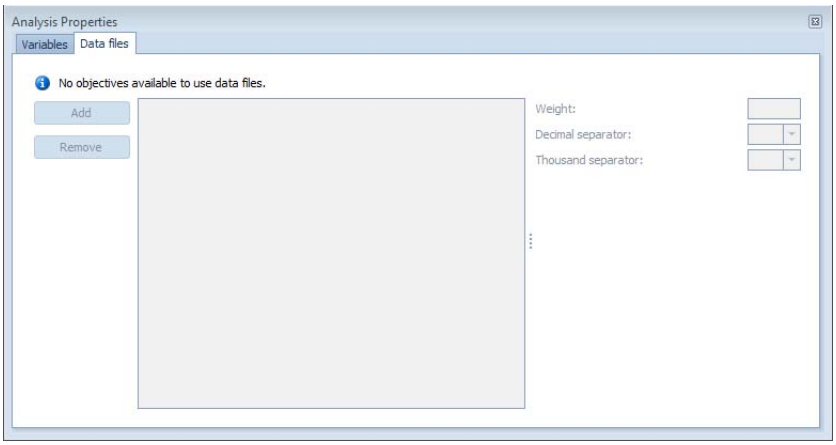


Figure 4.4 The Data files tab page of the Analysis Properties dialogue (Dynamic Simulation experiment)

Using the “Add” and “Remove” buttons data files can be added and removed (Figure 4.4). These data files will be scanned for reference time series in case either one of the Mean Difference, Maximum Difference or Theil's Inequality Coefficient criteria is enabled.

For each data file, the following properties can be specified:

- Weight
- Decimal separator

- Thousand separator

4.2 Local Sensitivity Analysis Experiment

The Local Sensitivity Analysis (LSA) Experiment allows for studying the sensitivity of ‘variables’ to small changes in ‘parameters’. These ‘variables’ can be any of the following model items:

- Algebraic variables
- Derived variables
- Output variables

On the other hand, ‘parameters’ can be any of the following model items:

- Manipulated variables
- Parameters
- Derived variables (more specifically: initial conditions)
- Input variables

4.2.1 Analysis Properties

The tab pages of the Analysis Properties window for the Local Sensitivity Analysis Experiment are described below and shown in Figure 4.5.

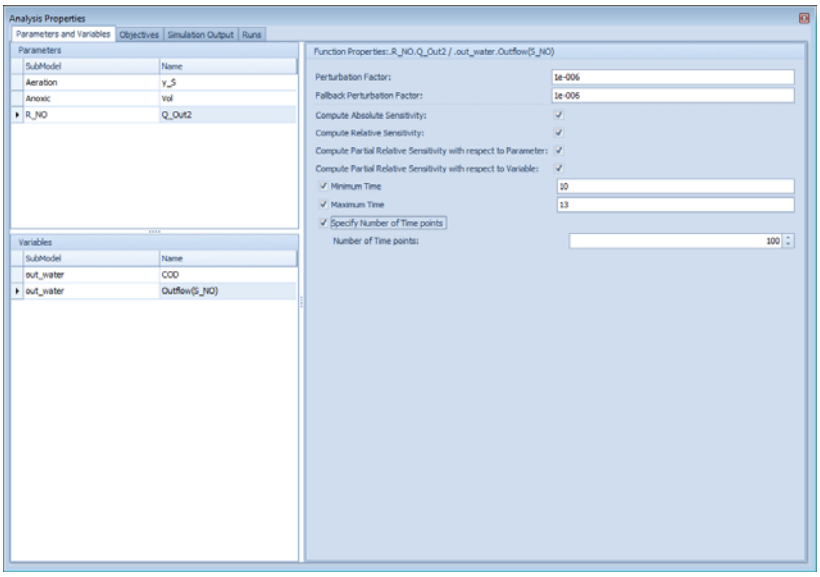


Figure 4.5 Parameters and Variables tab page for the LSA experiment

Parameters and Variables

- **Top left pane**:
  List of parameters that are to be used in the analysis. Items can be added to this list by drag-and-drop from the Block Details window, and can be deleted by pressing the Delete button.
- **Bottom left pane**:
  List of variables that are to be used in the analysis. Items can be added to this list by drag-and-drop from the Block Details window, and can be deleted by pressing the Delete button.
- **Right pane:**
  Settings for the parameter-variable combination selected in the top and bottom left panes.
  - Perturbation factor
  - Fallback Perturbation Factor
  - Compute Absolute Sensitivity
  - Compute Relative Sensitivity
  - Compute Partial Relative Sensitivity with respect to Parameter
  - Compute Partial Relative Sensitivity with respect to Variable
  - Minimum Time
  - Maximum Time
  - Specify Number of Time points
  - Number of Time points
  - Time Interval

Objectives

- Compute Backward Difference
- Compute Central Difference
- Compute Forward Difference

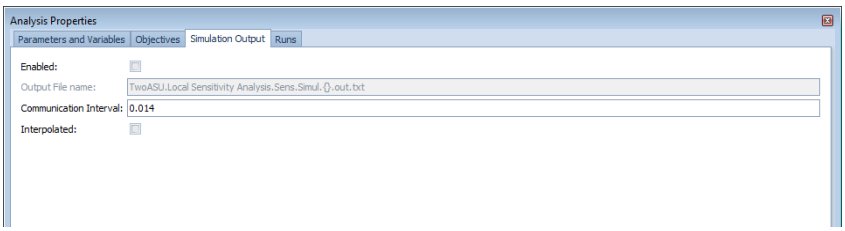


Figure 4.6 Simulation Output tab page for the LSA experiment

Simulation Output

Properties to be specified are the following (Figure 4.6):

- **Enabled**:
  Activates storage of simulation results in output files.
- **Output File name**:
  Output file name pattern. The {} character sequence will be substituted by the run number.
- **Communication Interval**:
  Time interval between the data points in the output files.
- **Interpolated**:
  Activates linear interpolation at communication time points.

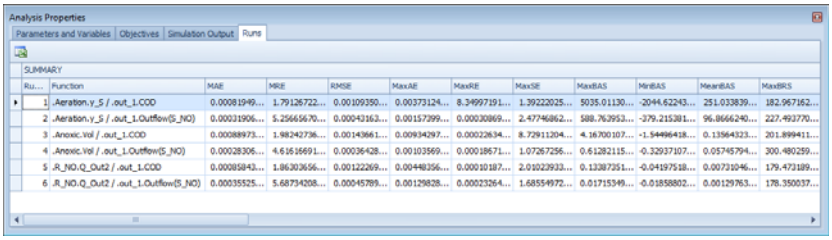


Figure 4.7 Runs tab page for the LSA experiment

Runs

The Runs table (Figure 4.7) is a matrix that gives an overview of the sensitivity functions that have been computed and a number of related aggregated values.

Atop the table are the following buttons:

- **Export**:
  Export the contents of the table to Excel (either as XLS or XLSX).
- **Print**:
  Allows for printing the Runs table.

The following quantities are computed (for each parameter/variable combination):

- **Errors**:
  - Mean Absolute Error (MAE)
  - Mean Relative Error (MRE)
  - Root Mean Squared Error (RMSE)
  - Maximum Absolute Error (MaxAE)
  - Maximum Relative Error (MaxRE)
  - Maximum Squared Error (MaxSE)
- **Sensitivities**:

- Backward Absolute Sensitivity:
  Maximum (MaxBAS)
  Minimum (MinBAS)
  Mean (MeanBAS)
- Backward Relative Sensitivity:
  Maximum (MaxBRS)
  Minimum (MinBRS)
  Mean (MeanBRS)
- Backward Partial Relative Sensitivity with respect to Variable:
  Maximum (MaxBPRSV)
  Minimum (MinBPRSV)
  Mean (MeanBPRSV)
- Backward Partial Relative Sensitivity with respect to Quantity:
  Maximum (MaxBPRSQ)
  Minimum (MinBPRSQ)
  Mean (MeanBPRSQ)
- Central Absolute Sensitivity:
  Maximum (MaxCAS)
  Minimum (MinCAS)
  Mean (MeanCAS)
- Central Relative Sensitivity:
  Maximum (MaxCRS)
  Minimum (MinCRS)
  Mean (MeanCRS)
- Central Partial Relative Sensitivity with respect to Variable:
  Maximum (MaxCPRSV)
  Minimum (MinCPRSV)
  Mean (MeanCPRSV)
- Central Partial Relative Sensitivity with respect to Quantity:
  Maximum (MaxCPRSQ)
  Minimum (MinCPRSQ)
  Mean (MeanCPRSQ)
- Forward Absolute Sensitivity:
  Maximum (MaxFAS)
  Minimum (MinFAS)
  Mean (MeanFAS)
- Forward Relative Sensitivity:
  Maximum (MaxFRS)
  Minimum (MinFRS)
  Mean (MeanFRS)
- Forward Partial Relative Sensitivity with respect to Variable:
  Maximum (MaxFPRSV)
  Minimum (MinFPRSV)
  Mean (MeanFPRSV)
- Forward Partial Relative Sensitivity with respect to Quantity:
  Maximum (MaxFPRSQ)
  Minimum (MinFPRSQ)
  Mean (MeanFPRSQ)

4.3 Global Sensitivity Analysis Experiment

The Global Sensitivity Analysis (GSA) Experiment will run a series of simulations on the basis of the same model, using different values for a selected set of ‘parameters’ for each simulation run. These ‘parameters’ can be either of the following model items:

- Manipulated variables
- Parameters
- Derived variables (more specifically: initial conditions)
- Input variables

On the basis of the simulated time series, time series criteria can be computed over a selected set of variables. Moreover, linear regression will be performed for each objective value in relation to the set of parameters.

Values are not determined for each parameter individually, but through a process that ensures equal spreading of parameter vectors within the parameter space (named Latin Hypercube Sampling).

4.3.1 Analysis Properties

The tab pages of the Analysis Properties window for the Global Sensitivity Analysis Experiment are described below and shown in (Figure 4.8).

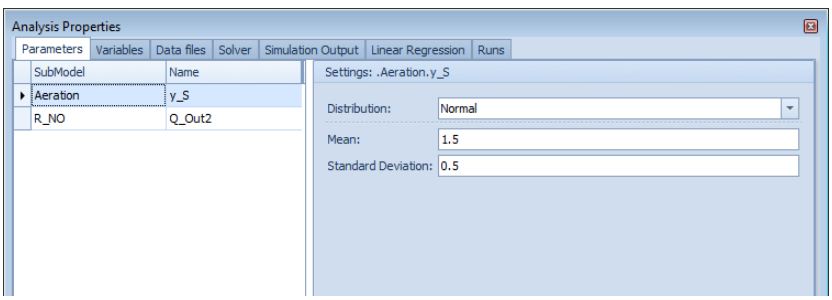


Figure 4.8 Parameters tab page for the GSA experiment

Parameters

- **Left pane**:
  List of parameters which will be assigned different values for each of the simulation runs. Items can be added to this list through drag-and-drop from the Block Details window, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  Settings for the parameter selected in the left pane.

– ***Distribution***: Statistical distribution that mimics the uncertainty on the actual value of the parameter.

a. *Uniform*: Uniform distribution. Additional properties to be specified are the following:
Lower bound
Upper bound

b. *Normal*: Normal (Gaussian) distribution. Additional properties to be specified are the following:
Mean
Standard Deviation

c. *LogNormal*: Lognormal distribution. Additional properties to be specified are the following:
Mean
Standard Deviation

d. *Exponential*: Exponential distribution. Additional properties to be specified are the following:
Rate

e. *Weibull*: Weibull distribution. Additional properties to be specified are the following:
Scale
Shape

f. *Triangular*: Triangular distribution. Additional properties to be specified are the following:
Lower Bound
Mode
Upper Bound

WEST - © DHI A/S

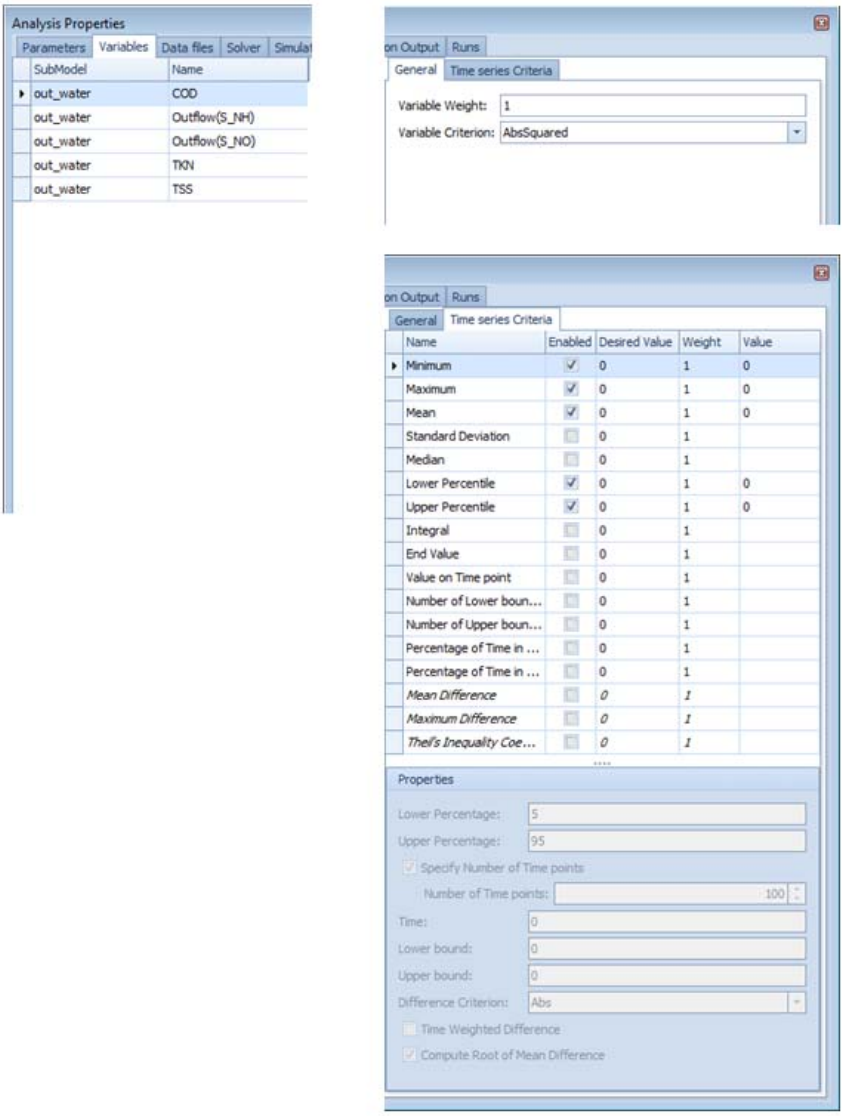


Figure 4.9 Variables tab page for the GSA experiment: variables list (left) and settings, General (top right) and Time series Criteria (bottom right) tabs

Variables

- **Left pane** (Figure 4.9):
  List of variables for which criteria are to be computed. Items can be added to this list through drag-and-drop from the Block Details pane, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  See Table 4.1, page 119.

Data Files

Using the “Add” and “Remove” buttons data files can be added and removed. These data files will be scanned for reference time series in case either one between the Mean Difference, Maximum Difference, Theil's Inequality Coefficient or End Value Difference time series criteria is enabled.

For each data file, the following properties can be specified:

- Weight
- Decimal separator
- Thousand separator

Solver

Two solver methods are available:

- **LHS**:
  Latin Hypercube Sampling
- **PR**:
  Pure random

Properties to be specified are the following:

- **Number of Shots**:
  Number of shots to be generated by the solver, which will determine the number of simulation runs
- **Seed**:
  Initial seed for the internal random number generator.

Simulation Output

Properties to be specified are the following (same as in Figure 4.6):

- **Enabled**:
  Activates storage of simulation results in output files.
- **Output File name**:
  Output file name pattern. The {} character sequence will be substituted by the run number.
- **Communication Interval**:
  Time interval between the data points in the output files.
- **Interpolated**:
  Activates linear interpolation at communication time points.

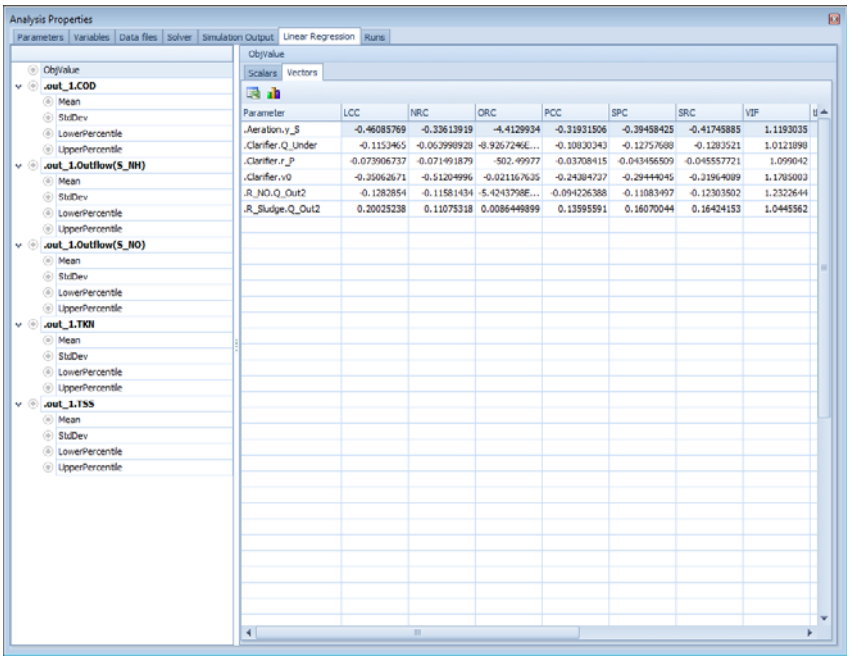


Figure 4.10 Linear Regression tab page for the GSA experiment

Linear Regression

- **Left pane**:
  Displays a structured list of objective values for which linear regression results are available.
- **Right pane**:
  Linear regression results for the objective selected in the left pane.
  - ***Scalars***:
    ESS: Error Sum of Squares
    F: F-statistics
    MSE: Mean Square Error
    MSR: Mean Square Regression
    MST: Mean Square Total
    RSS: Regression Sum of Squares
    TSS: Total Sum of Squares
    r2: Coefficient of Determination
    r2adj: Adjusted Coefficient of Determination

- ***Vectors***:
  LCC: Linear Correlation Coefficient
  NRC: Normalised Correlation Coefficient
  ORC: Ordinary Regression Coefficient
  PCC: Partial Correlation Coefficient
  SPC: Semi-partial Correlation Coefficient
  SRC: Standardised Regression Coefficient
  VIF: Variance Inflation Factor
  tLCC: t-statistics of LCC
  tPCC: t-statistics of PCC
  tSPC: t-statistics of SPC
  tSRC: t-statistics of SRC
  r: Residuals

Atop the table are the following buttons:

- **Export**:
  Export the contents of the table to Excel (either as XLS or XLSX).
- **Print**
  Allows for printing the table.
- **Plot** (only Vectors tab page):
  Allows for instantly displaying GSA-specific bar charts (cf. WEST Tutorial).

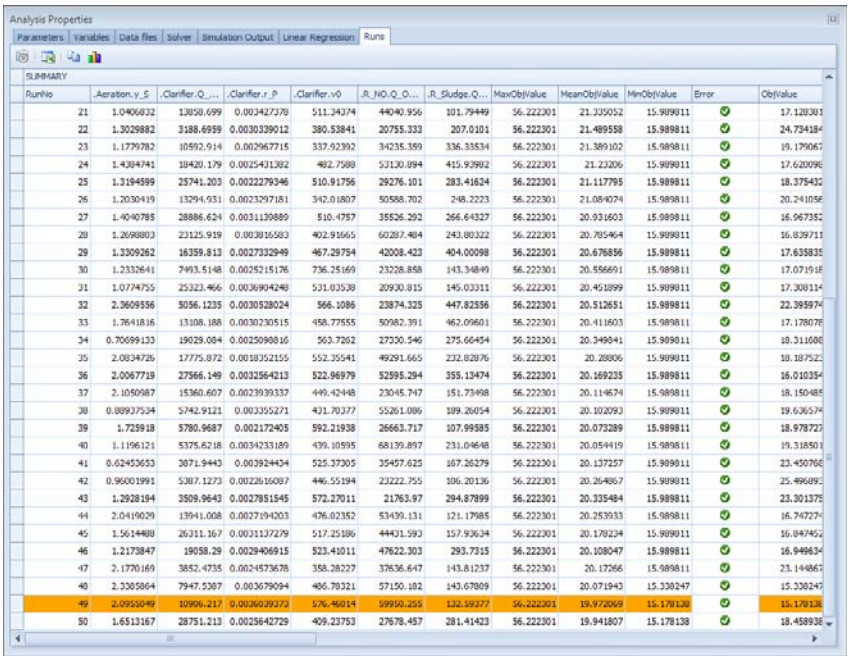


Figure 4.11 Runs tab page for the GSA experiment

Runs

The Runs table (Figure 4.11) is a matrix that gives an overview of the parameter values for each run, and the associated computed objective values (cf. time series criteria). The best run (i.e. the one with the lowest overall objective value) is highlighted.

Atop the table are the following buttons:

- **Generate**:
  Generate a new set of parameter values.
- **Export**:
  Export the contents of the table to Excel (either as XLS or XLSX).
- Print:
  Allows for printing the Runs table.
- Copy Values:
  Copy the parameter values of the selected run to the simulation experiment.
- **Run**:
  Allows for copying the parameter values and execute the simulation.
- **Plot**:
  Allow for instantly displaying histograms for any specified quantity relevant to the GSA experiment.

4.4 Parameter Estimation Experiment

The Parameter Estimation (PE) Experiment will run a series of simulations on the basis of the same model, using automatically generated values for a selected set of ‘parameters’ for each simulation run, in order to minimise an objective function.

These ‘parameters’ can be either of the following model items:

- Manipulated variables
- Parameters
- Derived variables (more specifically: initial conditions)
- Input variables

On the basis of the simulated time series, time series criteria can be computed on a selected set of variables, through the aggregation of data over time. The aggregation of these time series criteria is the objective function that is used during the optimisation (i.e. minimisation) process.

4.4.1 Analysis Properties

The tab pages of the Analysis Properties window for the Parameter Estimation Experiment are described below and shown in Figure 4.12.

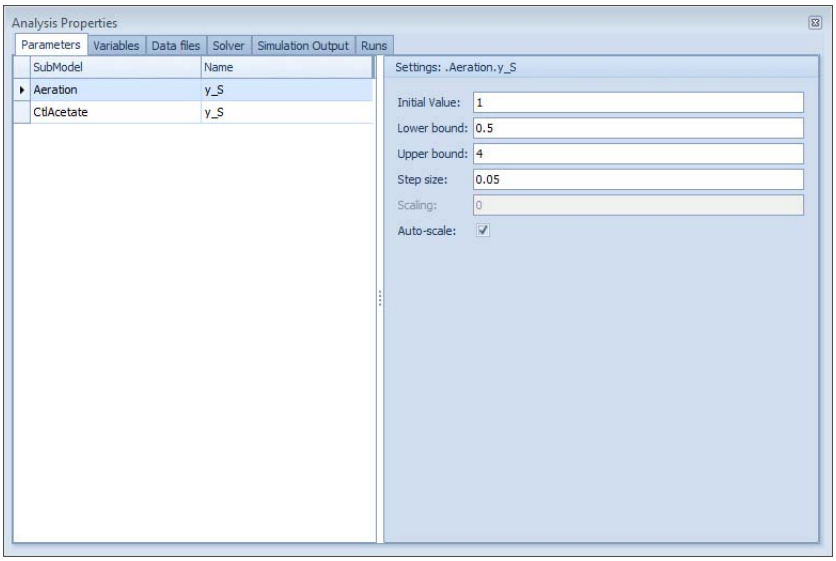


Figure 4.12 Parameters tab page for the PE experiment: parameters list (left) and settings (right)

Parameters

- **Left pane**:
  List of parameters which will be assigned different values for each of the simulation runs. Items can be added to this list through drag-and-drop from the Block Details window, and can be deleted by pressing the “Delete” button.
- **Right pane**:
  Settings for the parameter selected in the left pane.
  - Initial Value: Initial value to be used by optimiser
  - Lower Bound: Lower bound in case of constrained optimisation
  - Upper Bound: Upper bound in case of constrained optimisation
  - Step size: Size of the steps to be taken by the optimiser
  - Scaling: Scale factor to be used by the optimiser
  - Auto-scale: If switched on, Scaling is set to the Initial Value automatically (in order to scale to 1).

Variables

- **Left pane** (same as in Figure 4.9):
  List of variables for which criteria are to be computed. Items can be added to this list through drag-and-drop from the Block Details pane, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  See Table 4.1, page 119.

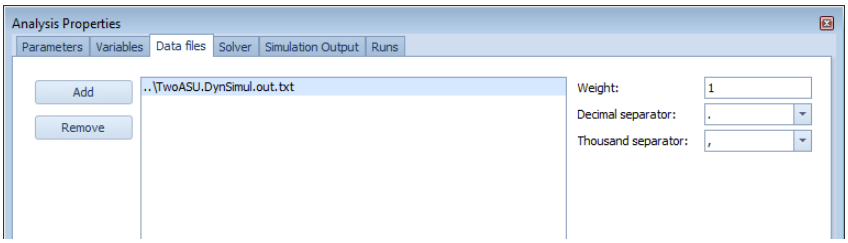


Figure 4.13 The Data files tab page for the PE experiment

Data Files

Using the “Add” and “Remove” buttons data files can be added and removed (Figure 4.13). These data files will be scanned for reference time series in case either one between the Mean Difference, Maximum Difference, Theil's Inequality Coefficient or End Value Difference criteria is enabled.

For each data file, the following properties can be specified:

- Weight
- Decimal separator
- Thousand separator

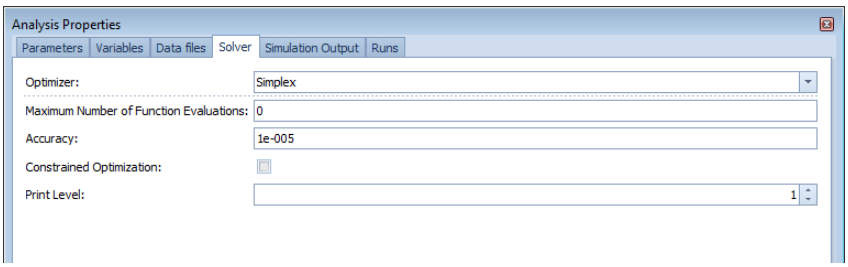


Figure 4.14 The Solver tab page for the PE experiment

Solver

The following optimisation methods are available (Figure 4.14):

- **Simplex**:
  Properties to be specified are the following:
  - Maximum Number of Function Evaluations
  - Accuracy
  - Constrained
  - Print Level

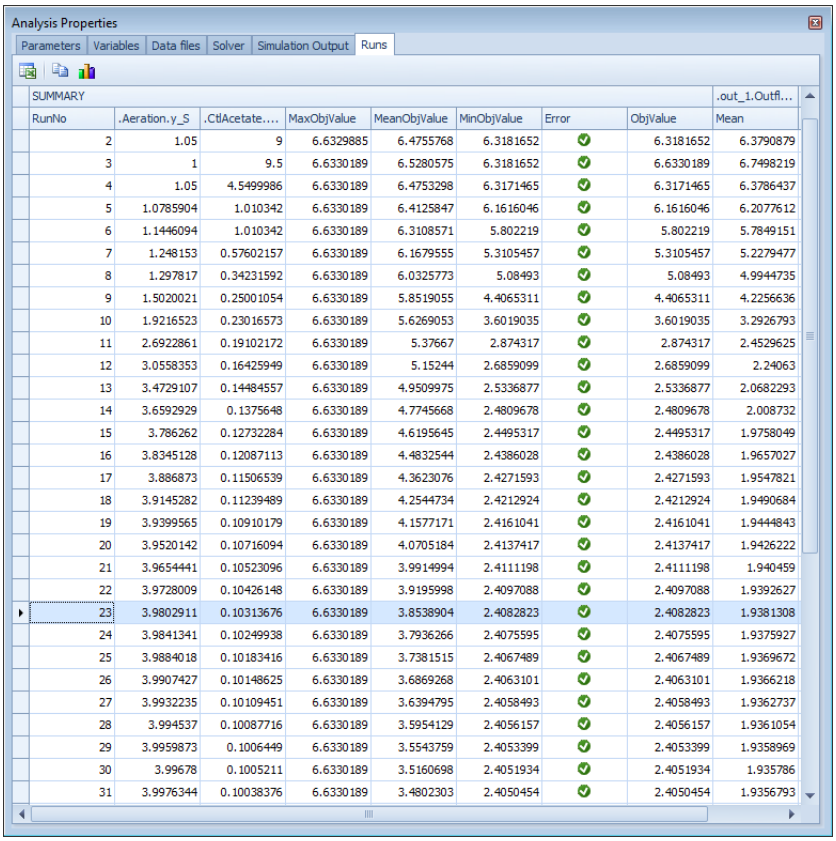
- **Praxis**:
  Properties to be specified are the following:
  - Maximum Number of Function Evaluations
  - Maximum Number of Stops
  - Accuracy
  - Maximum Step Size
  - Scale Bound
  - Constrained Optimisation
  - Ill-conditioned Problem
  - Covariance Matrix Calculation
  - Print Level
  - Seed for Random Number Generator

- **NLOpt**:
  Properties to be specified are the following:
  - Algorithm
  - Maximum Number of Function Evaluations
  - Constrained Optimization
  - Maximization of Objective Function
  - Maximum Time
  - Seed for Random Number Generator
  - Absolute Tolerance Objective Value
  - Absolute Tolerance Parameters
  - Relative Tolerance Objective Value
  - Relative Tolerance Parameters

Simulation Output

Properties to be specified are the following (same as Figure 4.6):

- **Enabled**:
  Activates storage of simulation results in output files.
- **Output File name**:
  Output file name pattern. The {} character sequence will be substituted by the run number.
- **Communication Interval**:
  Time interval between the data points in the output files.
- **Interpolated**:
  Activates linear interpolation at communication time points.



SUMMARY								
								.out_1.Outfl...
RunNo	.Aeration.y_S	.CtlAcetate....	MaxObjValue	MeanObjValue	MinObjValue	Error	ObjValue	Mean
2	1.05	9	6.6329885	6.4755768	6.3181652	✓	6.3181652	6.3790879
3	1	9.5	6.6330189	6.5280575	6.3181652	✓	6.6330189	6.7498219
4	1.05	4.5499986	6.6330189	6.4753298	6.3171465	✓	6.3171465	6.3786437
5	1.0785904	1.010342	6.6330189	6.4125847	6.1616046	✓	6.1616046	6.2077612
6	1.1446094	1.010342	6.6330189	6.3108571	5.802219	✓	5.802219	5.7849151
7	1.248153	0.57602157	6.6330189	6.1679555	5.3105457	✓	5.3105457	5.2279477
8	1.297817	0.34231592	6.6330189	6.0325773	5.08493	✓	5.08493	4.9944735
9	1.5020021	0.25001054	6.6330189	5.8519055	4.4065311	✓	4.4065311	4.2256636
10	1.9216523	0.23016573	6.6330189	5.6269053	3.6019035	✓	3.6019035	3.2926793
11	2.6922861	0.19102172	6.6330189	5.37667	2.874317	✓	2.874317	2.4529625
12	3.0558353	0.16425949	6.6330189	5.15244	2.6859099	✓	2.6859099	2.24063
13	3.4729107	0.14484557	6.6330189	4.9509975	2.5336877	✓	2.5336877	2.0682293
14	3.6592929	0.1375648	6.6330189	4.7745668	2.4809678	✓	2.4809678	2.008732
15	3.786262	0.12732284	6.6330189	4.6195645	2.4495317	✓	2.4495317	1.9758049
16	3.8345128	0.12087113	6.6330189	4.4832544	2.4386028	✓	2.4386028	1.9657027
17	3.886873	0.11506539	6.6330189	4.3623076	2.4271593	✓	2.4271593	1.9547821
18	3.9145282	0.11239489	6.6330189	4.2544734	2.4212924	✓	2.4212924	1.9490684
19	3.9399565	0.10910179	6.6330189	4.1577171	2.4161041	✓	2.4161041	1.9444843
20	3.9520142	0.10716094	6.6330189	4.0705184	2.4137417	✓	2.4137417	1.9426222
21	3.9654441	0.10523096	6.6330189	3.9914994	2.4111198	✓	2.4111198	1.940459
22	3.9728009	0.10426148	6.6330189	3.9195998	2.4097088	✓	2.4097088	1.9392627
23	3.9802911	0.10313676	6.6330189	3.8538904	2.4082823	✓	2.4082823	1.9381308
24	3.9841341	0.10249938	6.6330189	3.7936266	2.4075595	✓	2.4075595	1.9375927
25	3.9884018	0.10183416	6.6330189	3.7381515	2.4067489	✓	2.4067489	1.9369672
26	3.9907427	0.10148625	6.6330189	3.6869268	2.4063101	✓	2.4063101	1.9366218
27	3.9932235	0.10109451	6.6330189	3.6394795	2.4058493	✓	2.4058493	1.9362737
28	3.994537	0.10087716	6.6330189	3.5954129	2.4056157	✓	2.4056157	1.9361054
29	3.9959873	0.1006449	6.6330189	3.5543759	2.4053399	✓	2.4053399	1.9358969
30	3.99678	0.1005211	6.6330189	3.5160698	2.4051934	✓	2.4051934	1.935786
31	3.9976344	0.10038376	6.6330189	3.4802303	2.4050454	✓	2.4050454	1.9356793

Figure 4.15 Runs tab page for the PE experiment

Runs

The Runs table (Figure 4.15) is a matrix that gives an overview of the parameter values for each run, and the associated computed objective values (cf. time series criteria). The best run (i.e. the one with the lowest overall objective value) is highlighted.

Atop the table are the following buttons:

- **Export**:
  Export the contents of the table to Excel (either as XLS or XLSX).
- Print:
  Allows for printing the Runs table.
- Copy Values:
  Copy the parameter values of the selected run to the simulation experiment.
- **Run**:
  Allows for copying the parameter values and execute the simulation.

- **Plot**:
  Allow for instantly displaying histograms for any specified quantity relevant to the PE experiment.

4.5 Scenario Analysis Experiment

The Scenario Analysis (SA) Experiment will run a series of simulations on the basis of the same model, using different values for a selected set of 'parameters' for each simulation run. These 'parameters' can be either of the following model items:

- Manipulated variables
- Parameters
- Derived variables (more specifically: initial conditions)
- Input variables

On the basis of the simulated time series, time series criteria can be computed on a selected set of variables, through the aggregation of data over time.

4.5.1 Analysis Properties

The tab pages of the Analysis Properties window for the Scenario Analysis Experiment are described below and shown in Figure 4.16.

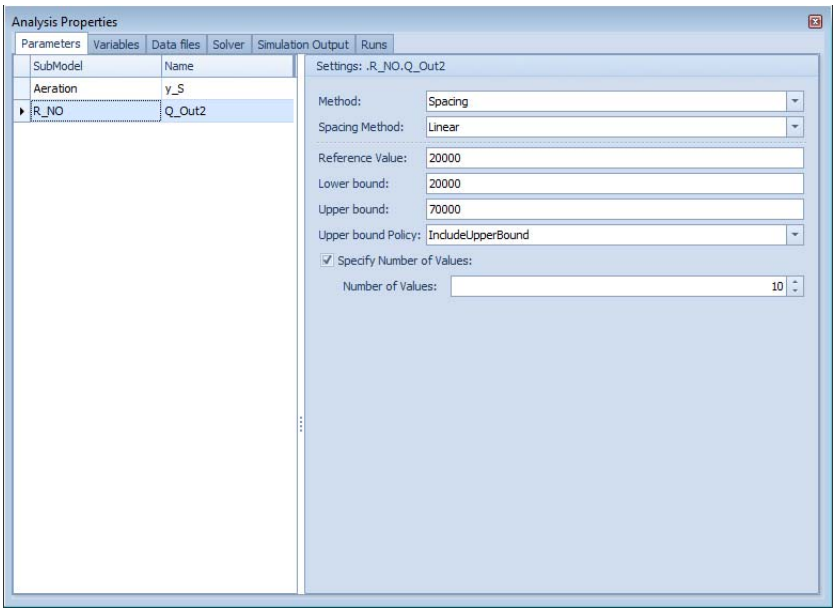


Figure 4.16 Parameters tab page for the SA experiment: parameters list (left) and settings (right)

Parameters

- **Left pane**:
  List of parameters which different values for each of the simulation runs will be assigned. Items can be added to this list through drag-and-drop from the Block Details window, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  Settings for the parameter selected in the left pane.
  - Method: Method to be used to determine the set of values for the parameter. The options are:

    a. ***Placement***: All values are to be listed up explicitly by the user. Properties to be specified are the following:
    - *Values*: Semicolon-separated list of values.
    - *Reference Value*: Value that is to be used as a reference. Only relevant for the Cross and Grid solvers (cf. below).

    b. ***Spacing***: Values are automatically generated through equidistant placement between a lower and upper bound. Properties to be specified are the following:
    - *Spacing Method*: Linear and Logarithmic
    - *Reference Value*: Value that is to be used as a reference. Only relevant for the Cross and Grid solvers (cf. below).
    - *Lower bound*

- *Upper bound*
- *Upper bound Policy*:
IncludeUpperBound: Include the Upper bound as specified as the last value in the list of values, even though it doesn't match the last computed value.
IncludeComputed: Include the last computed value as the last value in the list of values, even though it is higher than the Upper bound as specified.
Exclude: Exclude the Upper bound as specified from the list of values if it doesn't match the last computed value.
- *Specify Number of Values*: Switches between Number of Values and Interval.

c. ***Sampling***: Values are sampled from a statistical distribution. Properties to be specified are the following:
- *Distribution*: Statistical distribution
Uniform: Uniform distribution. Additional properties to be specified are the following: *Lower bound* and *Upper bound*
Normal: Normal (Gaussian) distribution. Additional Properties to be specified are the following: *Mean* and *Standard Deviation*
LogNormal: Lognormal distribution. Additional properties to be specified are the following: *Mean* and *Standard Deviation*
Exponential: Exponential distribution. Additional properties to be specified are the following: *Rate*
Weibull: Weibull distribution. Additional properties to be specified are the following: *Scale* and *Shape*
Triangular: Triangular distribution. Additional properties to be specified are the following: *Lower bound*, *Mode*, *Upper bound*
- *Type*: Continuous or Discrete
- *Reference Value*: Value that is to be used as a reference. Only relevant for the Cross and Grid solvers (cf. below).
- *Lower bound*
- *Upper bound*
- *Number of Values*

Variables

- **Left pane** (as in Figure 4.9):
List of variables for which criteria are to be computed. Items can be added to this list through drag-and-drop from the Block Details pane, and can be deleted by pressing the “Delete” button.

- **Right pane**:
See Table 4.1, page 119.

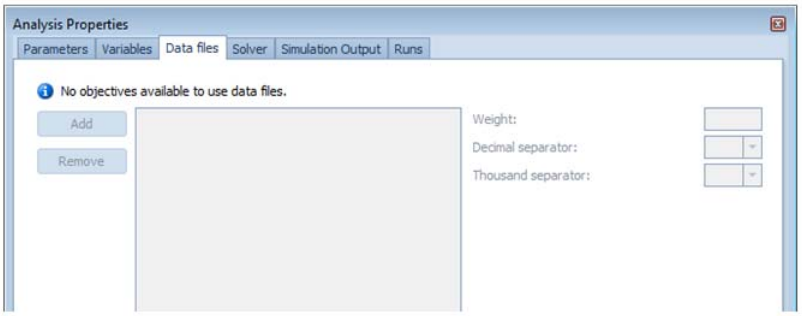


Figure 4.17 Data files tab page for the SA experiment

Data Files

Using the “Add” and “Remove” buttons data files can be added and removed (Figure 4.17). These data files will be scanned for reference time series in case either one between the Mean Difference, Maximum Difference, Theil's Inequality Coefficient or End Value Difference criteria is enabled. For each data file, the following properties can be specified:

- Weight
- Decimal separator
- Thousand separator

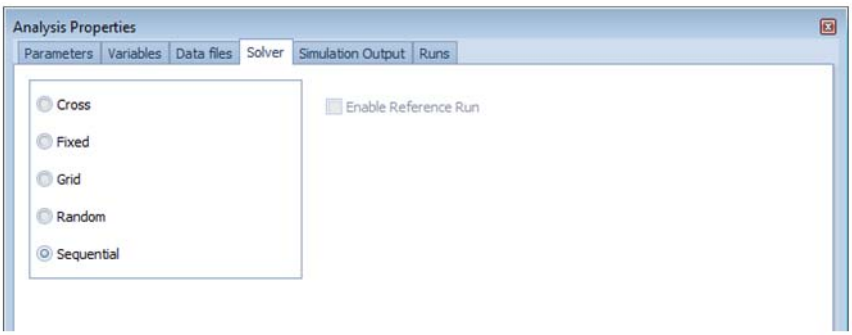


Figure 4.18 Solver tab page for the SA experiment

Solver

The following Scenario Analysis solvers can be used (Figure 4.18):

- **Sequential:**
  The Sequential solver simply steps through the list of variable value vector index combinations as it has been generated.

The total number of simulations that is run is given by the following expression:

$$N = \prod_{i=1}^{p} n_i$$

where:
p is the number of parameter
n_i is the number of values for the i-*th* parameter

- **Random**:
The Random solver is similar to the Sequential solver in the sense that it will eventually handle all variable value vector index combinations. However, the order in which this is done is determined by a random number generator with a uniform distribution.

- **Fixed**:
The Fixed solver does not run simulations for every variable value vector index combination.

  Instead it requires all variable value vectors to be of equal size and simply generates value vectors for each shot by taking elements from variable value vectors at the same position.

  The total number of simulations that is run is:

  $$N = n_i$$

- **Grid**:
The Grid Solver generates value vectors by stepping through the list of variable value vector index combinations. Subsequently, the list of value vectors is sorted according to their distance to a reference location.

  A user-supplied flag indicates whether the reference vector itself is also to be added to the value vector list. In case it is not, the number of value vectors, and hence the total number of simulation to be run, is again as in the Sequential case; otherwise the total number of shots is:

  $$N = 1 + \prod_{i=1}^{p} n_i$$

WEST - © DHI A/S

- **Cross**:
  Unlike the Grid solver, the Cross solver does not handle all possible combinations of variable values. In fact, it generates value vectors by starting from the reference vector and letting all elements of the reference vector, one after the other, vary using the values of the corresponding variable value vectors. The generated value vectors are sorted according to their distance to the reference vector.

  A user-supplied flag indicates whether the reference vector itself is to be added to the list. In case it is not, the total number of simulations to be run is given by the following expression:

$$N = \sum_{i=1}^{p} n_i$$

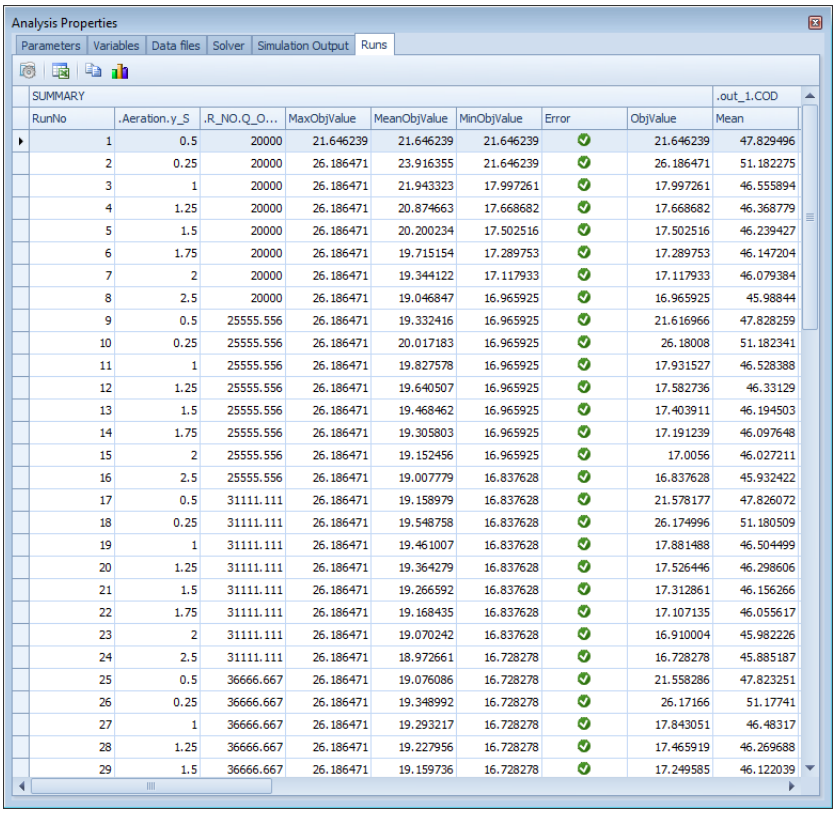
  In case the reference vector is prepended to the list, the total number of simulations to be run is:

$$N = 1 + \sum_{i=1}^{p} n_i$$

Simulation Output

Properties to be specified are the following (Figure 4.6):

- **Enabled**:
  Activates storage of simulation results in output files.
- **Output File name**:
  Output file name pattern. The {} character sequence will be substituted by the run number.
- **Communication Interval**:
  Time interval between the data points in the output files.
- **Interpolated**:
  Activates linear interpolation at communication time points.



SUMMARY								.out_1.COD
RunNo	.Aeration.y_S	.R_NO.Q_O...	MaxObjValue	MeanObjValue	MinObjValue	Error	ObjValue	Mean
1	0.5	20000	21.646239	21.646239	21.646239		21.646239	47.829496
2	0.25	20000	26.186471	23.916355	21.646239		26.186471	51.182275
3	1	20000	26.186471	21.943323	17.997261		17.997261	46.555894
4	1.25	20000	26.186471	20.874663	17.668682		17.668682	46.368779
5	1.5	20000	26.186471	20.200234	17.502516		17.502516	46.239427
6	1.75	20000	26.186471	19.715154	17.289753		17.289753	46.147204
7	2	20000	26.186471	19.344122	17.117933		17.117933	46.079384
8	2.5	20000	26.186471	19.046847	16.965925		16.965925	45.98844
9	0.5	25555.556	26.186471	19.332416	16.965925		21.616966	47.828259
10	0.25	25555.556	26.186471	20.017183	16.965925		26.18008	51.182341
11	1	25555.556	26.186471	19.827578	16.965925		17.931527	46.528388
12	1.25	25555.556	26.186471	19.640507	16.965925		17.582736	46.33129
13	1.5	25555.556	26.186471	19.468462	16.965925		17.403911	46.194503
14	1.75	25555.556	26.186471	19.305803	16.965925		17.191239	46.097648
15	2	25555.556	26.186471	19.152456	16.965925		17.0056	46.027211
16	2.5	25555.556	26.186471	19.007779	16.837628		16.837628	45.932422
17	0.5	31111.111	26.186471	19.158979	16.837628		21.578177	47.826072
18	0.25	31111.111	26.186471	19.548758	16.837628		26.174996	51.180509
19	1	31111.111	26.186471	19.461007	16.837628		17.881488	46.504499
20	1.25	31111.111	26.186471	19.364279	16.837628		17.526446	46.298606
21	1.5	31111.111	26.186471	19.266592	16.837628		17.312861	46.156266
22	1.75	31111.111	26.186471	19.168435	16.837628		17.107135	46.055617
23	2	31111.111	26.186471	19.070242	16.837628		16.910004	45.982226
24	2.5	31111.111	26.186471	18.972661	16.728278		16.728278	45.885187
25	0.5	36666.667	26.186471	19.076086	16.728278		21.558286	47.823251
26	0.25	36666.667	26.186471	19.348992	16.728278		26.17166	51.17741
27	1	36666.667	26.186471	19.293217	16.728278		17.843051	46.48317
28	1.25	36666.667	26.186471	19.227956	16.728278		17.465919	46.269688
29	1.5	36666.667	26.186471	19.159736	16.728278		17.249585	46.122039

Figure 4.19 Runs tab page for the SA experiment

Runs

The Runs table (Figure 4.19) is a matrix that gives an overview of the parameter values for each run, and the associated computed objective values (cf. time series criteria). The best run (i.e. the one with the lowest overall objective value) is highlighted.

Atop the table are the following buttons:

- Generate:
  Generate a new set of parameter values. This will only have an effect when at least one parameter's method is set to sampling.
- **Export**:
  Export the contents of the table to Excel (either as XLS or XLSX).
- Print:
  Allows for printing the Runs table.

- Copy Values:
  Copy the parameter values of the selected run to the simulation experiment.
- **Run**:
  Allows for copying the parameter values and execute the simulation.
- **Plot**:
  Allow for instantly displaying histograms for any specified quantity relevant to the SA experiment.

4.6 Uncertainty Analysis Experiment

The Uncertainty Analysis (UA) Experiment will run a series of simulations on the basis of the same model, using different values for a selected set of 'parameters' for each simulation run. These 'parameters' can be either of the following model items:

- Manipulated variables
- Parameters
- Derived variables (more specifically: initial conditions)
- Input variables

On the basis of the simulated time series, 4 types of criteria can be computed over a selected set of variables:

- **Time series Criteria**:
  Aggregation of simulated data across time for a particular simulation run.
- **Run Criteria**:
  Aggregation of simulated data across runs for a particular time point.
- **BVDF (Bound Value Duration Frequency) Criteria**:
  Evaluation of the violation of a set of bounds for each simulation run.
- **Classification Criteria**:
  Classification of the simulated values of a run according to a number of predefined classes.

Values are not determined for each parameter individually, but through a process that ensures equal spreading of parameter vectors within the parameter space (named Latin Hypercube Sampling).

4.6.1 Analysis Properties

The tab pages of the Analysis Properties window for the Uncertainty Analysis Experiment are described below.

Parameters

- **Left pane** (same as in Figure 4.8):
  List of parameters which will be assigned different values for each of the simulation runs. Items can be added to this list through drag-and-drop from the Block Details window, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  Settings for the parameter selected in the left pane.
  - ***Distribution***: Statistical distribution that mimics the uncertainty on the actual value of the parameter.
    - *Uniform*: Uniform distribution. Additional properties to be specified are: Lower- and Upper bound
    - *Normal*: Normal (Gaussian) distribution. Additional properties to be specified are: Mean and Standard Deviation
    - *LogNormal*: Lognormal distribution. Additional properties to be specified are: Mean and Standard Deviation
    - *Exponential*: Exponential distribution. Additional property to be specified is: Rate
    - *Weibull*: Weibull distribution. Additional properties to be specified are: Scale and Shape
    - *Triangular*: Triangular distribution. Additional properties to be specified are: Lower Bound, Mode and Upper Bound.

Variables

- **Left pane** (same as in Figure 4.9):
  List of variables for which criteria are to be computed. Items can be added to this list through drag-and-drop from the Block Details pane, and can be deleted by pressing the “Delete” button.

- **Right pane**:
  See Table 4.1, page 119.

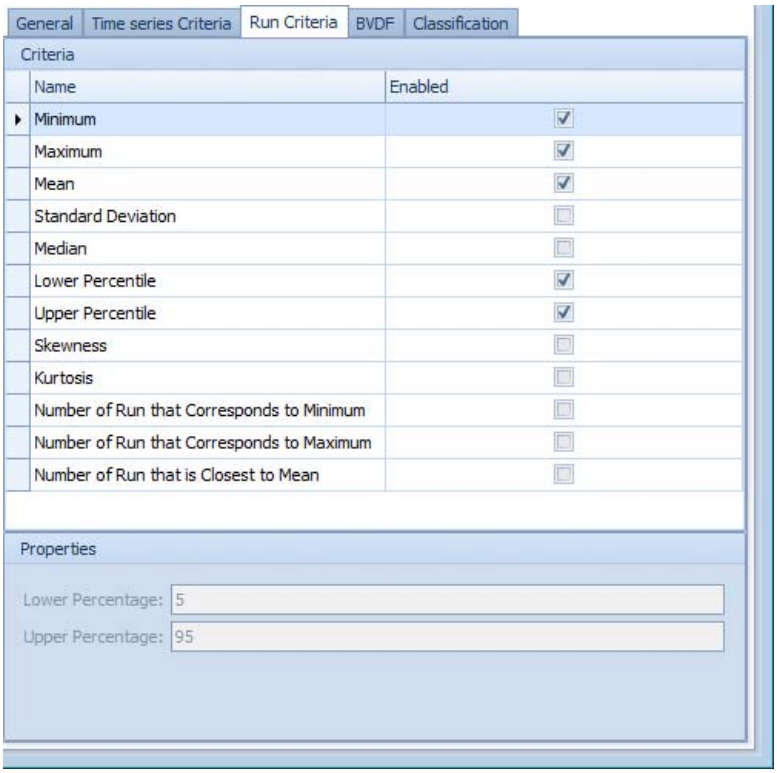


Figure 4.20 Run Criteria tab page for the UA experiment

– ***Run Criteria*** *(Figure 4.20)*:
a. *Minimum*: Minimum of values across all runs.
b. *Maximum*: Maximum of values across all runs.
c. *Mean*: Mean of values across all runs.
d. *Standard Deviation*: Standard deviation of values across all runs.
e. *Median*: Median of values across all runs.
f. *Lower Percentile*: Lower percentile of values across all runs. Additional property to be specified: Lower Percentage
g.*Upper Percentile*: Upper percentile of values across all runs. Additional property to be specified: Upper Percentage
h. *Lower Percentile 2*: Second lower percentile
i.*Upper Percentile 2*: Second upper percentile
j. *Skewness*: Skewness of the values across all runs
k. *Kurtosis*: Kurtosis of the values across all runs.
l. *Moment*
m. *Number of Run that Corresponds to Minimum*
n. *Number of Run that Corresponds to Maximum*
o. *Number of Run that is Closest to Mean*

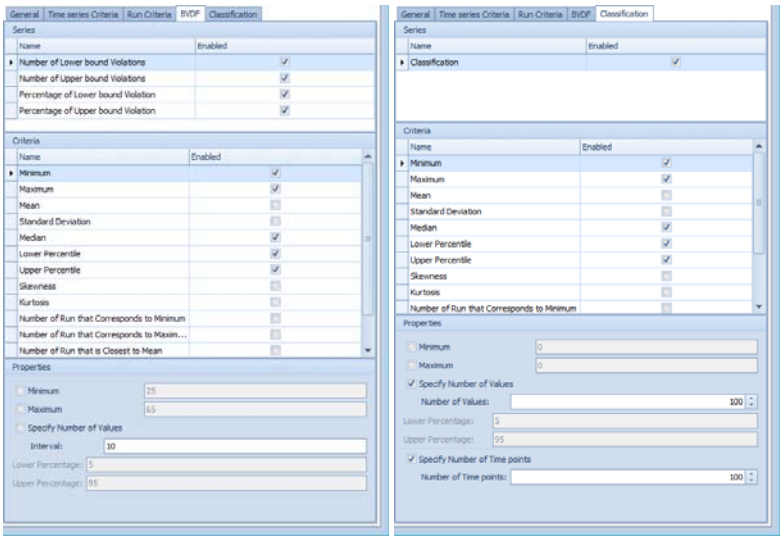


Figure 4.21 BVDF (left) and Classification (left) tab pages for the UA experiment

– **BVDF (Figure 4.21)**:
a. *Series*
- Number of Lower bound Violations
- Number of Upper bound Violations
- Percentage of Lower bound Violation
- Percentage of Upper bound Violation

b. *Criteria*
- *Minimum*: Minimum of values across all runs.
- *Maximum*: Maximum of values across all runs.
- *Mean*: Mean of value across all runs.
- *Standard Deviation*: Standard deviation of values across all runs.
- *Median*: Median of values across all runs.
- *Lower Percentile*: Lower percentile of values across all runs. Additional property to be specified: Lower Percentage
- *Upper Percentile*: Upper percentile of values across all runs. Additional property to be specified: Upper Percentage
- *Lower Percentile* 2: Second lower percentile
- *Upper Percentile 2*: Second upper percentile
- *Skewness*: Skewness of the values across all runs.
- *Kurtosis*: Kurtosis of the values across all runs.
- *Moment*
- *Number of Run that Corresponds to Minimum*
- *Number of Run that Corresponds to Maximum*
- *Number of Run that is Closest to Mean*

– **Classification (Figure 4.21)**:
a. *Series*
- *Classification*

b. *Criteria*
- *Minimum*: Minimum of values across all runs.
- *Maximum*: Maximum of values across all runs.
- *Mean*: Mean of value across all runs.
- *Standard Deviation*: Standard deviation of values across all runs.
- *Median*: Median of values across all runs.
- *Lower Percentile*: Lower percentile of values across all runs. Additional property to be specified: Lower Percentage
- *Upper Percentile*: Upper percentile of values across all runs. Additional property to be specified: Upper Percentage
- *Lower Percentile 2:* Second lower percentile
- *Upper Percentile* 2: Second upper percentile
- *Skewness*: Skewness of the values across all runs.
- *Kurtosis*: Kurtosis of the values across all runs.
- *Moment*
- Number of Run that Corresponds to Minimum
- Number of Run that Corresponds to Maximum
- Number of Run that is Closest to Mean

Data Files

Using the “Add” and “Remove” buttons data files can be added and removed. These data files will be scanned for reference time series in case either one between the Mean Difference, Maximum Difference, Theil's Inequality Coefficient or End Value Difference time series criteria is enabled.

For each data file, the following properties can be specified:

- Weight
- Decimal separator
- Thousand separator

Solver

Two solver methods are available:

- **LHS**:
  Latin Hypercube Sampling
- **PR**:
  Pure random

Properties to be specified are the following:

- **Number of Shots**:
  Number of shots to be generated by the solver, which will determine the number of simulation runs
- **Seed**:
  Initial seed for the internal random number generator.

Simulation Output

Properties to be specified are the following (same as in Figure 4.6):

- **Enabled**:
  Activates storage of simulation results in output files.
- **Output File name**:
  Output file name pattern. The {} character sequence will be substituted by the run number.
- **Communication Interval**:
  Time interval between the data points in the output files.
- **Interpolated**:
  Activates linear interpolation at communication time points.

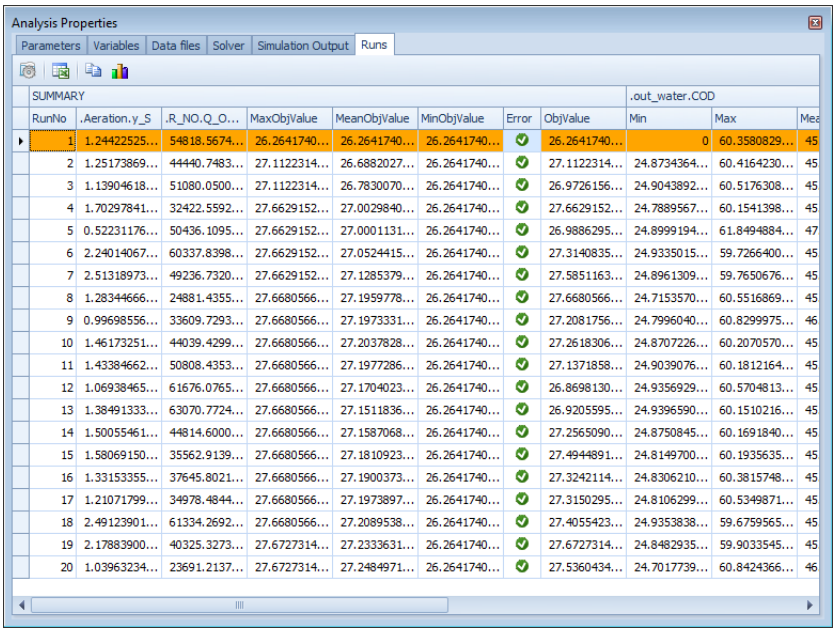


Figure 4.22 Runs tab page for the UA experiment

Runs

The Runs table (Figure 4.22) is a matrix that gives an overview of the parameter values for each run, and the associated computed objective values (cf. time series criteria). The best run (i.e. the one with the lowest overall objective value) is highlighted.

Atop the table are the following buttons:

- Generate:
  Generate a new set of parameter values.

- **Export**:
  Export the contents of the table to Excel (either as XLS or XLSX).
- Print:
  Allows for printing the Runs table.
- Copy Values:
  Copy the parameter values of the selected run to the simulation experiment.
- **Run**:
  Allows for copying the parameter values and execute the simulation.
- **Plot**:
  Allows for instantly displaying one of a set of frequently required plot windows related to BVDF and Classification criteria:
  - ***BVDF***:
    a. *BVDF Type*:
    Number of Lower Bound Violations
    Number of Upper Bound Violations
    Percentage Lower bound Violations
    Percentage Upper bound Violations

    b. *Specification*:
    1 Quantity - 1 Run
    1 Quantity - All Runs
    All Quantities - 1 Run
    Quantity Aggregation

    c. *Quantity*: Only relevant for
    1 Quantity - 1 Run, and
    1 Quantity - All Runs

    d. *Run Number*: Only relevant for
    1 Quantity - 1 Run, and
    All Quantities - 1 Run
  - ***Classification***:
    a. *Specification*:
    1 Quantity - 1 Run
    1 Quantity - All Runs
    All Quantities - 1 Run
    Quantity Aggregation

    b. *Quantity*: Only relevant for
    1 Quantity - 1 Run, and
    1 Quantity - All Runs

    c. *Run Number*: Only relevant for
    1 Quantity - 1 Run, and
    All Quantities - 1 Run

WEST - © DHI A/S

5 Use of WEST

This Chapter explains how to use WEST in practice. It consists of a series of units, each covering a specific topic and developed as a small tutorial that will take you from starting up a New Project in WEST to running a simulation.

For additional practical information on the use of WEST, refer to the Tutorial's guide.

5.1 How to Manage Projects

A Project in WEST consists of the following elements:

- **a Plant Layout** - the graphical representation of the Plant Model
- **a *base* experiment** - effectively an Objective Evaluation experiment which comprises a Steady-State and a Dynamic simulation
- **one *advanced* virtual experiment** (optional) - which is built on top of either the Steady-State or the Dynamic simulation
- any number of Dashboard Sheets
- **two input files**, for every Input Block of type Wastewater (Figure 2.9)

5.1.1 Create a New Project

There are two ways to create a New Project in WEST, i.e. from scratch or from a Template.

Create a New Project From Scratch

In the left frame of the Getting Started window (Figure 2.3), select the item Templates and then the folder All.

In the middle area, double-click on the card "Create blank Project" (or click on the button "Create" in the right frame under the blank Preview area). This will start up the "New Project" dialog, described below.

Fill in the relevant information for the New Project and confirm by clicking on the "OK" button.

Create a New Project From a Template

In the left frame of the Getting Started window (Figure 2.3), select the item Templates and then the folder All.

In the middle area, double-click on the card corresponding to the desired template (or click on the button "Create" in the right frame under the Preview area). This will start up the "New Project" dialog, described below.

Fill in the relevant information for the New Project and confirm by clicking on the “OK” button.

Templates are meant to provide a starting point to build more complex Plant Layouts; so only the key parameters are set (typically: kLa) and few sample plots are provided. In order to run a simulation, all relevant parameters will need to be set and the Influent and Effluent Models created (by going through the Influent and the Effluent Tools respectively).

The following properties are to be set in the New Project dialog window (Figure 5.1):

- **Name**:
  The name of the Project. By default, it is set to ‘ProjectX’, X being a progressive ID automatically assigned by WEST.
- **Location**:
  The location of the project folder. By default, it is set to the 'WEST Projects folder' as specified in the WEST Options (cf. Advanced). Click on the “Browse” button to select a different location.
- **Description**:
  A description for the Project (optional).
- **Block Library**:
  The Block Library to be used to compose the Plant Layout.
- **Instance**:
  An instance of the model library, which includes a selection of the ASM model and a declaration of the number of layers, e.g. for the clarifier models.

For the Name of a Project and the Location, a valid and unique combination is to be chosen to allow a new Project directory to be created.

The “Instance” drop-down can be expanded to show the available Instances and the corresponding Description.

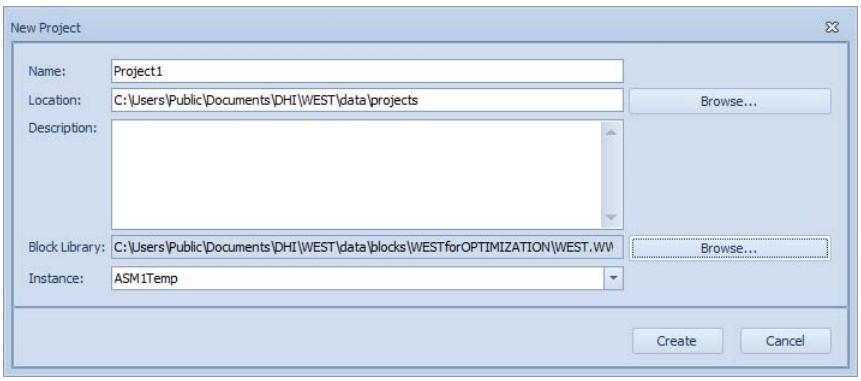


Figure 5.1 Input the preliminary information and settings for the New Project

5.1.2 Open an Existing Project

In the left frame of the Getting Started window (Figure 2.3), select the item “Projects” and then the folder “Recently used” or “All”: the first option will display in the middle area the recently used projects; the latter, all the projects stored in the Projects folder (as specified in the WEST Options, Advanced).

In the middle area, double-click on the card (or click on the button “Open” in the right frame under the Preview area).

5.1.3 Open a Sample

WEST contains sample Projects that are meant to illustrate its functionalities and that are briefly summarised in Table 5.1.

In the left frame of the Getting Started window (Figure 2.3), select the item “Samples” and then the folder “All”.

In the middle area, double-click on the card corresponding to the desired sample (or click on the button “Create” in the right frame under the Preview area).

Table 5.1 Sample Projects in WEST

BSM1_OL_ASM1	Benchmark test case no.1 (BSM1), with open loop (OL) scenario, for ASM1. The plant consists of a 5-compartment nitrogen removing activated sludge system: two anoxic tanks followed by three aerobic tanks and a secondary clarifier, in a basic pre-denitrification configuration. More information in Copp (2002).
BSM1_OL_ASMG2d	Variant of "BSM1_OL" for ASMG2d. The plant consists of 3 stages: a 2-compartments anaerobic tank, 2-compartment anoxic tank and a 3-compartments aerobic tanks followed by a secondary clarifier.
BSM2_OL_ASMG1	Benchmark test case no.2 (BSM2), with open loop (OL) scenario, for ASMG1. Simulation under steady-state and 609-day dynamic conditions. More information in Guo and Vanrolleghem (2014) and Nopens et al. (2010).
Fractionator_ASM1	Influent fractionator for ASM1. Probabilistic interpretation of measurements as compositions in terms of ASM1 components

Table 5.1 Sample Projects in WEST

Galindo_OL	Implementation of the Galindo WWTP, with open loop (OL) scenario. The Galindo WWTP is designed for C and N removal and handles a daily average wastewater flow 345600 m3/day. The WWTP is divided into six identical lines and the configuration in each of them is able to be operated as one of two alternative configurations: Regeneration-Denitrification-Nitrification (RDN) and Denitrification-Regeneration-Denitrification-Nitrification (DRDN).
Galindo_CL	Variant of the Galindo WWTP, with closed loop (CL) scenario i.e. advanced automatic control strategies.
Orbal	Model of the Orbal nutrient removal plant in Athens, Georgia, USA. The model is calibrated based on nitrate, oxygen, ammonium nitrogen and phosphate measurements.
OUR_MSL	Calibration of K_S and mu_max, based on OUR measurements.
TestCase_CEPT	Test case to demonstrate the use of the CEPT model.
TwoASU	Basic activated sludge system for Carbon and Nitrogen removal, consisting of two compartments, one anoxic and one aerated, followed by a secondary clarifier. Aeration in the second compartment is controlled. The influent file is a standard 30-day municipal type influent, derived from the benchmark exercise. The TwoASU sample is fully worked out in the Tutorial, Section 2.
TwoASU_Animation	Variant of the "TwoASU" sample, demonstrating the Animation feature: - Line Thickness is proportional to the flow rate - Line Colour is associated with the concentration of X_S - Labels show ammonia (S_NH) concentration - An Alert is set on the solids (TSS) concentration in the effluent for values exceeding an upper limit (15 g/m3)
Umbilo	Umbilo WWTP, South Africa. Calibrated model (data year 2003).

5.1.4 Close a Project

The current Project can be closed (without shutting down the application), by clicking on the “Close” button in the Application menu.

5.2 How to Create a Plant Layout

The Plant Layout is the graphical representation of the Plant Model that can be set up on the Layout Sheet by assembling Blocks from the Block Library and establishing connections between them.

5.2.1 Place Blocks on the Layout Sheet

The first step in creating the Plant Layout (and Model) that will further be used for the dynamic simulation consists in placing the relevant Blocks on the Layout Sheet.

This is done by dragging the icon of the Block from the Block Library to the Layout Sheet as illustrated in Figure 5.2.

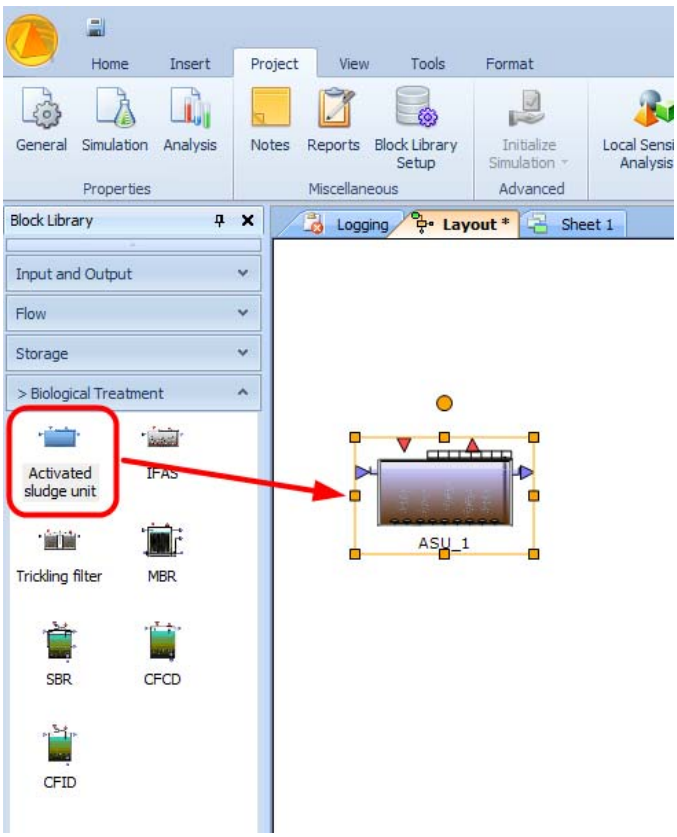


Figure 5.2 Drag and drop a Block Icon from the Block Library to the Layout Sheet

5.2.2 Assign a Block to a Layer

A Block can be assigned to a specific Layer (or to more than one Layer) by enabling the target Layer prior to dropping the Block on the canvas.
A Block can also be moved from one Layer to another by clicking on the ellipsis button next to the Layers property in the Properties pane of that Block and selecting the desired Layer(s) in the Layers Picker dialog (cf. Figure 5.3).

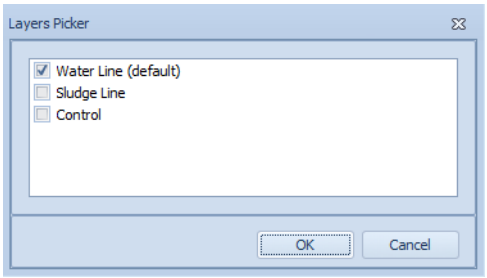


Figure 5.3 Layers Picker dialog to re-assign a Block to a Layer

5.2.3 Rename a Block

A Block has two Properties pertaining to its name:

- **DisplayName**:
  this is the label of the icon, as it appears on the Layout Sheet
- **Name**:
  this is the *internal* name that uniquely identifies the Block and that appears in the Model Explorer pane. Because it is used internally by WEST, only letters and digits can be used for a Block's Name (e.g. no spaces)

The (Display) Name of a Block can be changed by selecting the Block and:

- editing the DisplayName property in the Properties pane; or
- right-clicking on the Block in the Layout Sheet, choosing "Edit Label" from the Context menu (Figure 5.4) and confirming, by clicking outside the label.

The Edit Label menu item is only enabled when the whole Block is selected.

The DisplayName and the Name are synchronised by default that is: changes to one are immediately reflected to the other one.

This behaviour can be changed through the SynchronizeNames property of the Block in the Properties pane.

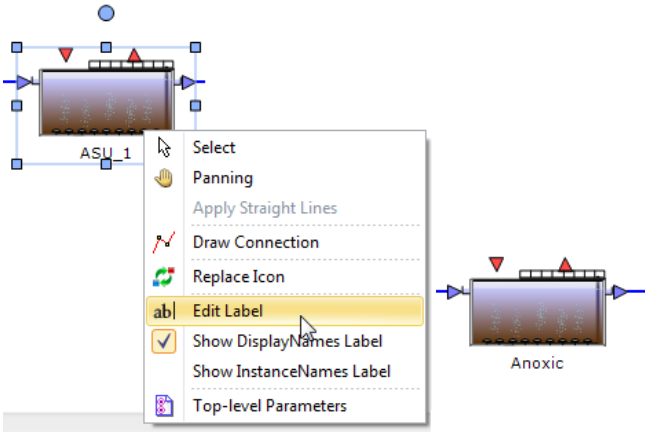


Figure 5.4 Rename a Block

5.2.4 Resize a Block

To resize a Block, proceed as follows:

- Select the Block in the Layout Sheet: a grey rectangle will be displayed around it with eight handles on the perimeter (light blue dots: resizing handles) and a larger external handle (dark blue dot: rotation handle)
- Hover the mouse over one of the handles until the mouse pointer changes into a two-headed arrow (Figure 5.5-a)
- Drag the resizing handle to reach the desired size of the Block (Figure 5.5-b)
- Release the mouse (Figure 5.5-c)

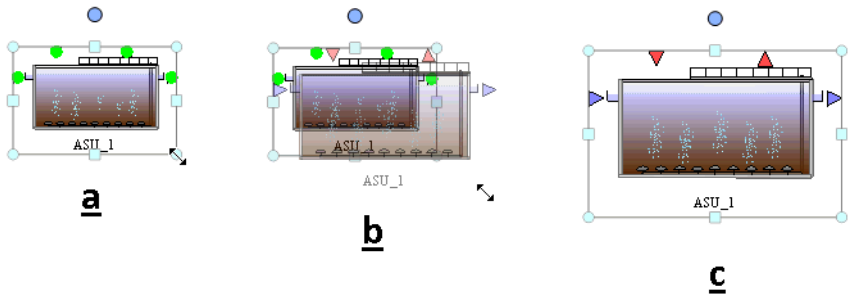


Figure 5.5 Resizing a Block

5.2.5 Rotate a Block

Two rotation modes are supported in WEST, i.e. **free** and **fixed rotation**.

Free Rotation of a Block

To *freely* rotate a Block, proceed as follows:

- Select the Block: a grey rectangle will be displayed around it with eight handles on the perimeter (light blue dots: resizing handles) and a larger external handle (dark blue dot: rotation handle)
- Hover the mouse over the rotation handle until the mouse pointer changes into a rounded arrow (Figure 5.6-a)
- Drag the selection handle to reach the desired rotation angle of the Block (Figure 5.6-b)
- Release the mouse (Figure 5.6-c)

The Block Icon or the Label can be rotated independently, by first selecting the Block, then the Icon or the Label and then proceeding as described.

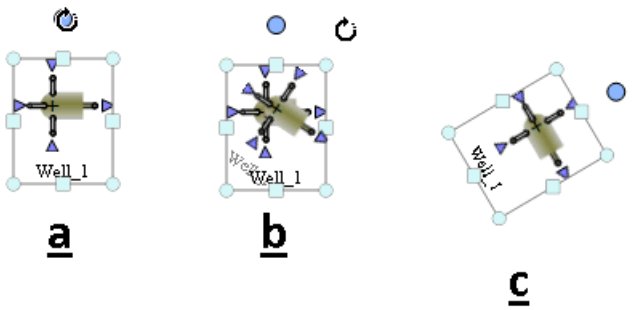


Figure 5.6 Rotating a Block: free rotation

Fixed Rotation of a Block

To rotate a Block by a fixed rotation angle, i.e. 90°, 180° or 270°, select the Block and click on the appropriate button in the “Format” Menu (Figure 5.7).

The Block Icon or the Label can be rotated independently, by first selecting the Block, then the Icon or the Label and then proceeding as described.

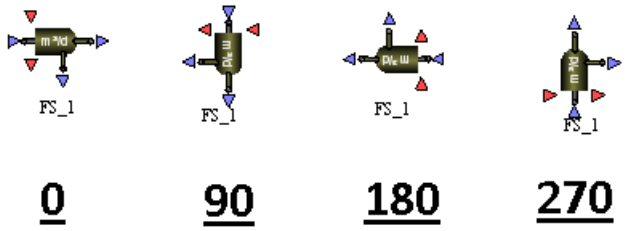


Figure 5.7 Rotating a Block: fixed rotation

5.2.6 Flip a Block

A third movement for a Block is supported in WEST, i.e. flipping (or: mirroring).

To mirror a Block, either around a vertical or a horizontal axis, select the Block and click on the appropriate button in the “Format” menu (Figure 5.8).

The Block Icon or the Label can be rotated independently, by first selecting the Block, then the Icon or the Label and then proceeding as described.

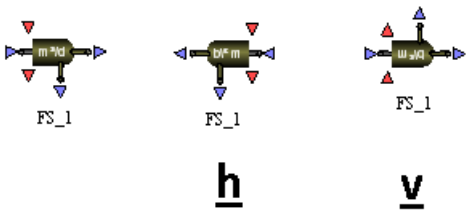


Figure 5.8 Flipping a Block

5.2.7 Replace the Icon of a Block

The graphical representation of a Block (its Icon) can be replaced in the Layout Sheet, by selecting the Block, activating the Context menu (right-click) and choosing Replace Icon. The dialog window shown in Figure 5.9 will be displayed.

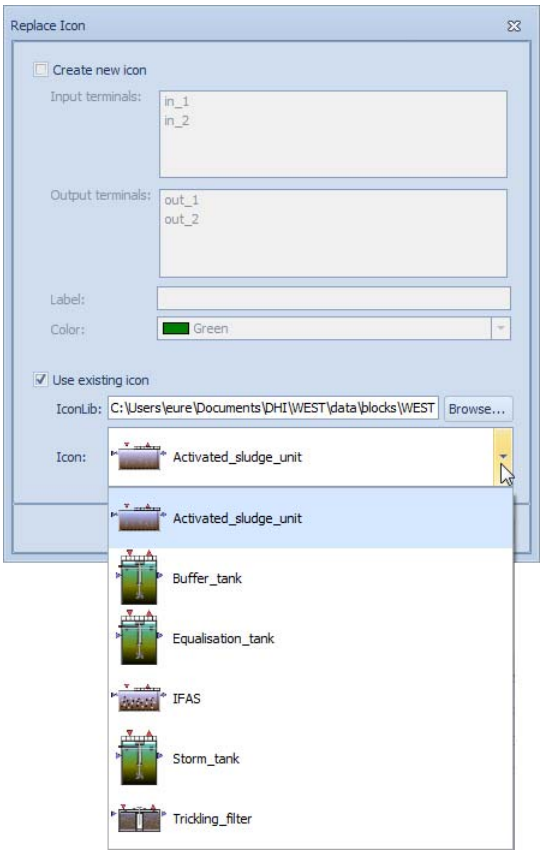


Figure 5.9 Replace Icon dialog to replace a Block's Icon

Available options are:

- to create a new icon: a square-shaped icon with the same number of input and output Terminals as the original, of a given colour and, optionally, with a given label will replace the current Icon;
- to use a compatible icon (in terms of number and type of Terminals) from a given IconLib.

When a Block's Icon is replaced by an 'existing' icon, nothing changes to the underlying Model associated with the Block.

5.2.8 Replacing a Block (and underlying model)

Dragging a Block from the Block Library onto an existing Block in the Layout (Figure 5.10, Top) results in effectively replacing not only its visual representation, but also the underlying model. If the interfaces of the two Block (and models) match, the pre-existing connections are maintained (Figure 5.10, Bottom).

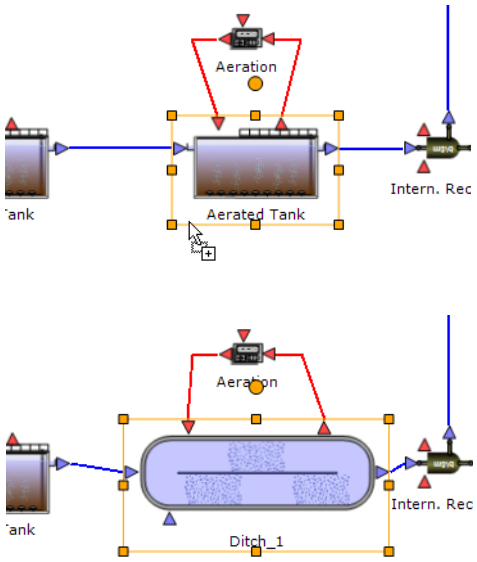


Figure 5.10 Automatic replacement of a Block and its associated mode. Before replacement (top), after replacement (bottom)

5.2.9 Connect two Blocks

Blocks are to be connected by graphically establishing link between the outgoing terminal of the block upstream and the ingoing terminal of the block downstream. Only Terminals of the same type can be connected.

- To create a connection between two Blocks, proceed as follows:
- Select “Polyline” (or “Orthogonal”) in the Insert | Connections menu (or in the Context menu).
- Hover over the first Block (the connection points will be highlighted) and click on the connection point:
  - (“Polyline” connection) Release, extend the line that is generated until it reaches the ingoing connection point of the second Block and click to terminate the connection; or
  - (“Orthogonal” connection) hold the mouse down, extend the line that is generated until it reaches the ingoing connection point of the second Block and release to terminate the connection.
- (only for “Polyline” connection) The direction of a connection can be changed by placing an anchor:
  - while extending the connection from the start terminal to the end terminal, click on the drawing board; every click corresponds to an anchor being inserted and the direction of the connection can change; or

– by clicking on an existing connection while holding the “Ctrl + Shift” key down

To exit the connection mode, right-click and choose the Select option on the context menu; or go to the Home | Editing menu and choose the Select option.

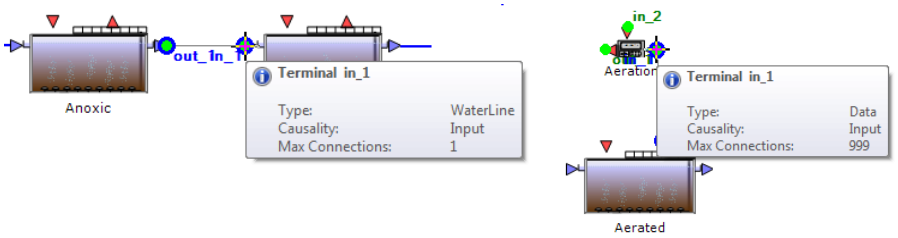


Figure 5.11 Creating a connection between two Terminals: mass flux- (left) and data terminals (right)

Once the connection is established, it is necessary to specify which quantity is transferred across the connection, i.e. the Interface Link.

In most cases, there is a single quantity that can be transferred - so WEST can automatically set the Interface Link. In other cases, generally involving a Control Block, the user is to select the appropriate quantities, because the in-tank sensors expose several measurements (e.g. DO, NH4, NO3, TSS) and most Blocks have more than one Manipulated Variables (control handles).

If this is the case, the Interface Link dialog pops up as the connection can be completed (Figure 5.12): select the appropriate From and To interface variables from the drop-down menus.

By double-clicking on a Connection, the Interface Links dialog pops up and an existing link may be modified.

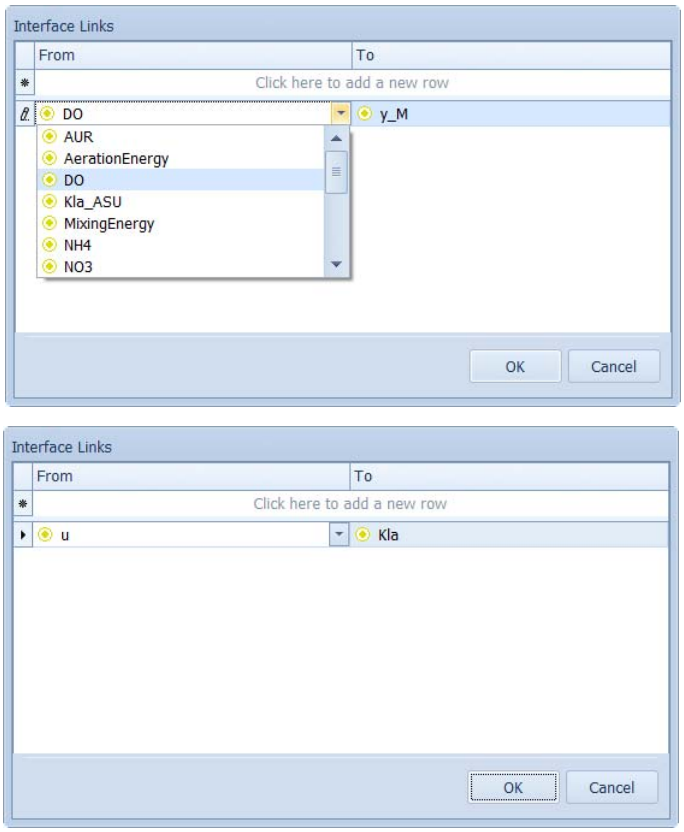


Figure 5.12 Specifying the Interface Links for the connection with a Controller Block: incoming data connection (top) and outgoing control data connection (bottom)

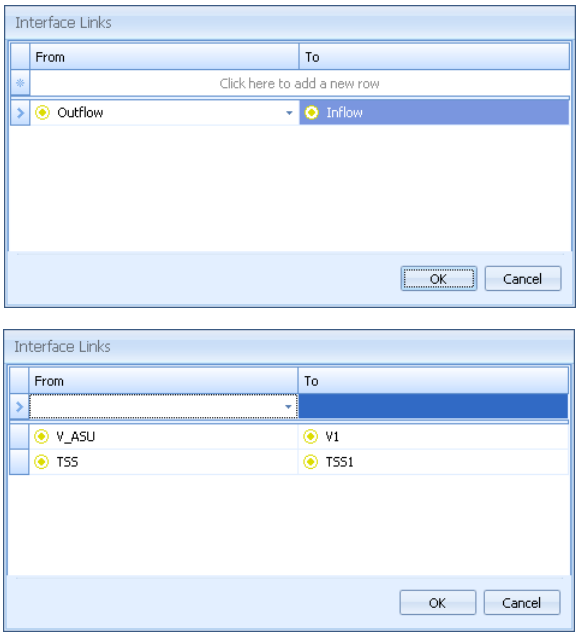


Figure 5.13 Dialog window to set (or review) the type of connection between two Blocks: mass flux connection (top) and data connection (bottom)

5.2.10 Align Blocks

Another common action related to the appearance of the Plant Layout is the alignment of the Blocks.

Alignment of two (or more) Blocks can be made with reference to their Icon or to their Terminals, by selecting the target Blocks and one of the following menu items from the “Format” menu:

- **Horizontal Terminals**:
  the Blocks are aligned *horizontally* with reference to their terminal, i.e. along a horizontal connection.
- **Vertical Terminals**:
  the Blocks are aligned *vertically* with reference to their terminal, i.e. along a vertical connection.
- **Top**:
  the Blocks are aligned to the top with reference to their whole Block Icon (incl. the Label).
- **Middle**:
  the Blocks are aligned to the middle (horizontal median axis) with reference to the whole Block Icon (incl. the Label).

- **Bottom**:
  the Blocks are aligned to the bottom with reference to the whole Block Icon (incl. the Label).
- **Left**:
  the Blocks are aligned to the left with reference to the whole Block Icon (incl. the Label).
- **Center**:
  the Blocks are aligned to the centre (vertical median axis) with reference to the whole Block Icon (incl. the Label).
- **Right**:
  the Blocks are aligned to the right with reference to the whole Block Icon (incl. the Label).

Furthermore, Blocks can be dragged freely on the Layout Sheet or moved in discrete steps by using the arrow keys on the keyboard; holding the “Shift” key down while clicking on an arrow key will result in discrete steps of 1 pixel.

Several Blocks can be selected by either drawing a selection rectangle around them or by clicking on each one individually while holding the “Ctrl” key down.

5.2.11 Change the Model Selection of a Block

In WEST, a Block may be associated with more than just one model.

A model other than the default one can be assigned to a Block as follows:

- Select the Block.
- In the Properties window, browse to the ClassName property and choose the desired model from the drop down list.

In WESTforDESIGN, only the Activated Sludge Unit, the SBR, the CFID (Continuous Flow Intermittent Decant) and the CFCD (Continuous Flow Continuous Decant) are associated with several models which account for different mixing conditions (completely mixed vs. plug flow) or for different operating schedules (i.e. number of reaction periods).

Another way to change a model is by dragging-dropping a new Block onto an existing one (see Replacing a Block (and underlying model), page 162).

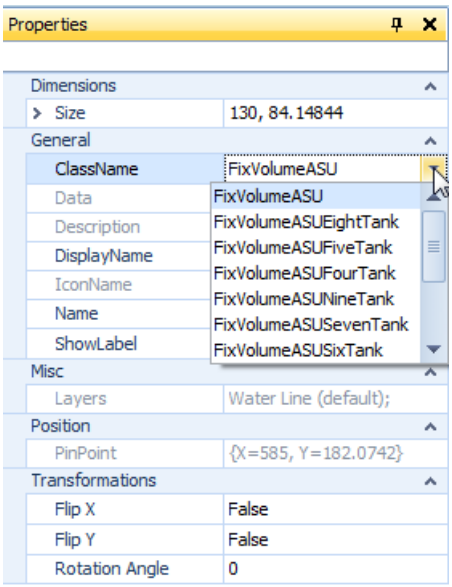


Figure 5.14 The Properties window of the ASU Block, showing the available models

5.2.12 Define Top-Level Parameters

Top-Level Parameters (Section 3.2.3) can be created manually, i.e. by defining their links to the respective Model Parameters; or by using the Easy Create dialog.

Manually Create Top-Level Parameters

- Right click anywhere in the Layout Sheet (do not select any specific object on the Sheet) and choose “Top-Level Parameters” in the context menu.
- In the Parameters dialog window (Figure 5.15), enter the parameter name and properties (cf. ‘Properties of Top-Level Parameters’ on page 169).
- Click “OK” to confirm and close the dialog window.

Easy Create Top-Level Parameters

- Select one or more Blocks in the Layout Sheet.
- The Block Details window will display the parameters sub-set that is common to the selected Blocks: select one or more of these parameters.
- Click “Easy Create Parameters” on the toolbar of the window.
- All relevant properties will be filled up in the Parameters dialog window (Figure 5.15): modify the parameter properties, if necessary.
- Click “OK” to confirm and close the dialog window.

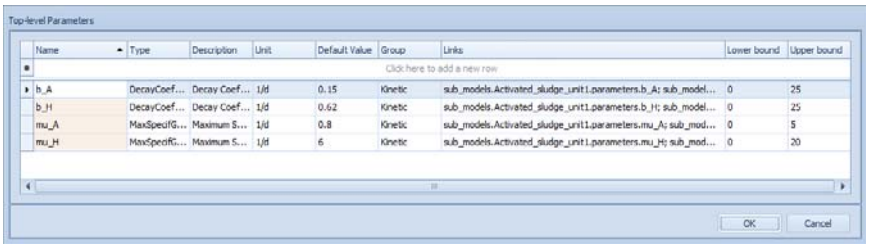


Figure 5.15 The top-level parameters dialog window, with a set of newly created top-level parameters

Properties of Top-Level Parameters

A Top-Level parameter has the following properties:

- **Name**:
  name of the Top-Level Parameter.
- **Type**:
  parameter type (the type carries information regarding the units, the range, i.e. upper and lower bounds, etc.
- **Description**:
  (optional) description of the Parameter.
- **Unit**:
  measuring unit of the Parameter.
- **Default Value**:
  default value proposed for the Parameter.
- **Group**:
  group that the Parameter is set to belong to and that will be used in the Block Details window to group parameters in sub-categories.
- **Links**:
  sub-model parameters that the Parameter is linked to.
- Lower Bound
- Upper Bound

5.2.13 Define Scalar Top-Level Interface Variables

Scalar Top-Level Interface Variables (Section 3.2.4) are to be defined manually, by selecting the corresponding item in the Context menu and providing a name and a set of properties of the interface variable (Figure 5.16).

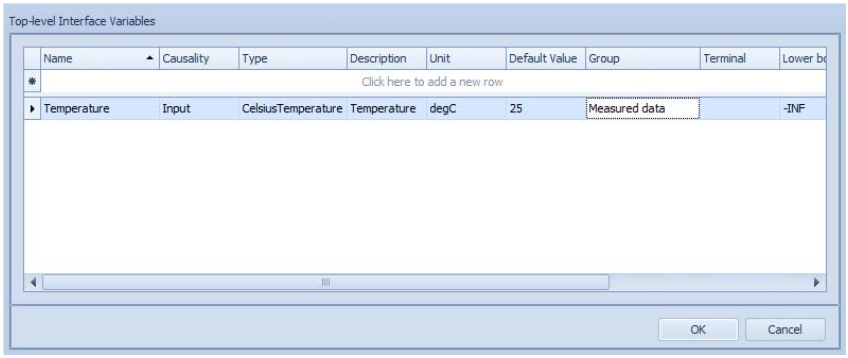


Figure 5.16 The top-level interface variables dialog window, with a newly created variable

Properties of Scalar Top-Level Interface Variables

- **Name**:
  Name of the variable.
- **Causality**:
  Causality: “Input” (e.g. if it is to be associated to a Data Input Block) or “Output” (e.g. if it is to be associated to a Data Output Block).
- **Type**:
  Data type: not mandatory (default is “Real”) but useful to allow for automatic unit conversion.
- **Description** (optional)
- **Unit**:
  Not mandatory (default is unitless) but useful to allow for automatic unit conversion.
- **Default Value**:
  Not mandatory (default is 0) but useful to allow for automatic unit conversion.
- **Group**:
  Not mandatory but necessary to have variables grouped in the Block Details pane.
- **Terminal**:
  Not mandatory but necessary to expose the variable through a terminal, if the plant model is used as coupled model.
- **Lower Bound**:
  Default is “-INF”.
- **Upper Bound**:
  Default is “+INF”.
- **Manipulated**:
  Indicates whether the variable can be manipulated in the scope of a coupled model.

- **Favorite**:
  Indicates whether the variable is to be displayed in the Block Summary pane, in the scope of a coupled model.

5.2.14 Define Calculator Variables

A Calculator Variable can be created as follows:

- select the Project Model (by either selecting the top-most node of the Model Explorer or clicking anywhere in the Layout)
- fill in the relevant information in the Calculator Variable dialog window that will be displayed
- click “OK” to confirm and close the dialog window

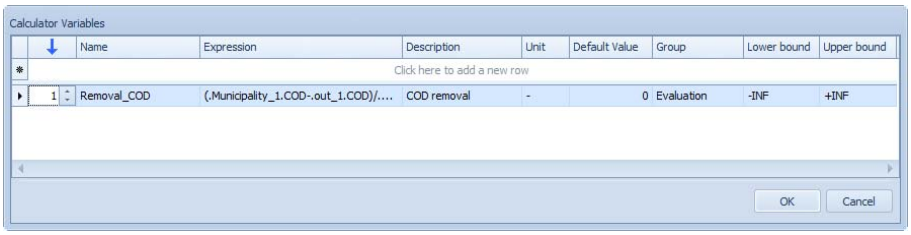


Figure 5.17 The Calculator Variable dialog window, with a newly created calculator variable

Properties of Calculator Variables

A Calculator Variable has the following properties:

- **Name**
  name of the Calculator Variable.

- **Expression**
  Insertion of quantity names is facilitated through a dropdown that is activated by typing “.”: the naming adopted throughout WEST applies, i.e. objects (parameters or variables) identifiers are to be prepended by the Block Name (internal name!), as follows:

  .<NameOfBlock>.<NameOfObject>

- **Description**
  (optional) description of the Variable

- **Unit**
  (optional) measuring unit of the Variable; can be selected from a drop-down list of available units

- **Default Value**
  Not mandatory (default is 0) but useful to allow for automatic unit conversion

- **Group**
  (optional) group that the Variable is set to belong to and that will be used in the Block Details window to group variables in sub-categories

- **Lowerbound**:
  lower bound for the Variable (default is -INF); a computed value smaller than the lower bound will be displayed in red in the Block Details pane (and will issue a Warning message in the Logging pane)

- **Upperbound**:
  upper bound for the Variable (default is +INF); a computed value greater than the upper bound will be displayed in red in the Block Details pane (and will issue a Warning message in the Logging pane)

5.2.15 How To Set Up a Controller

Model Parameters of the class Manipulated Variables can be dynamically altered during the simulation and therefore linked to a Control or to a Timer Block.

The basic set-up of a control *loop* requires that:

- a *measurement* is fed to the Control Block, through a Sensor Block or through a sensor variable that is available from within a specific Block (and exposed by its outgoing Data Terminal, cf. Terminals);
- the *signal* of the Control Block is used to alter the appropriate Manipulated Variable of the Block to be controlled (which is exposed by its incoming Data Terminal, cf. Terminals).

As any other Block, a Control Block can be controlled by another Control Block by receiving an external signal that is to be linked to one of the Manipulated Variables of the controller's model. For more details, refer to ‘Options to Control Model Parameters’ in the Tutorial’s guide.

5.3 How to Manage Dashboard Sheets

Dashboard Sheets act as containers for Input and Output widgets respectively; however, the same Sheet may contain both types of widgets.

Dashboard Sheets can be added to and removed from the Project.

Input (output) widgets can be added to, removed from and re-arranged within a Sheet; and can be moved to another Sheet through the Organize button in View menu.

5.3.1 Add or Delete a Sheet

A new Sheet can be added to the Project as follows:

- Choose “Add Sheet” in the Insert | Sheets menu
- Enter a name for the new Sheet and confirm with “OK”

A Sheet can be removed from the Project through the “Remove Sheet” button of the Insert Sheets menu.

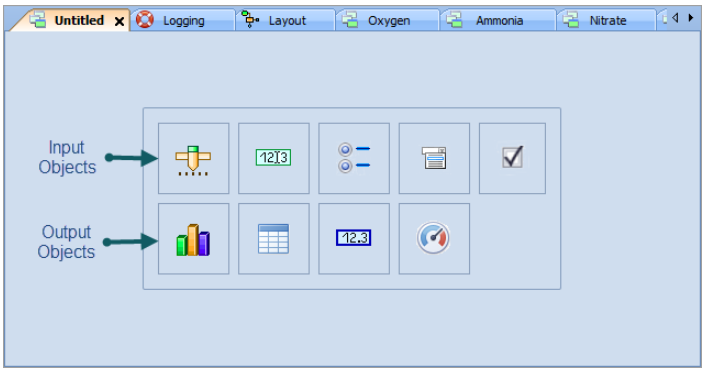


Figure 5.18 Create a new Sheet

5.3.2 Add or Delete an Input or Output Object

Input and output *objects* can be added to newly created Sheets and to Sheets that contain other input/output objects.

In the first instance, i.e. an empty Sheet, click on the respective button in the frame in the middle of the Sheet (Figure 5.18); in the latter case, click on the respective button in the Insert Menu.

Input and output *objects* can be removed from a Sheet, by clicking on the blue arrow in the top right corner of the object and selecting ‘Remove <nameoftheobject>’.

5.3.3 Arrange Input and Output Objects Within a Sheet

By default, input and output objects are simply stacked on a Sheet (Figure 5.19) but their relative position and size can be changed as follows:

- Click on “Design Mode” in the View | Sheets menu.
- Move the Customisation dialog window out of the way, if necessary, to access the view shown in Figure 5.20.

- *objects* may be repositioned, relatively to the other objects in the same 'Sheet', by dragging their Title bar (Figure 5.21);
- *objects* may also be resized, by moving their edges (Figure 5.22)

- Quit the Design Mode by clicking the close button of the Customization dialog window.

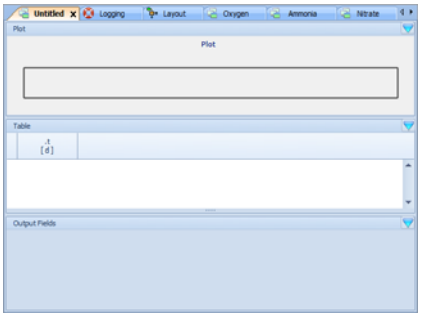


Figure 5.19 New plot, table and field items, stacked on a new Sheet

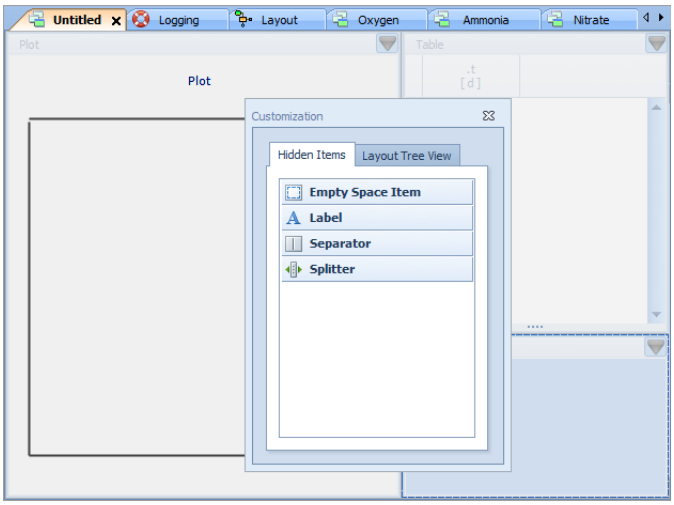


Figure 5.20 New plot, table and field items, re-arranged on a new Sheet in 'Design Mode'

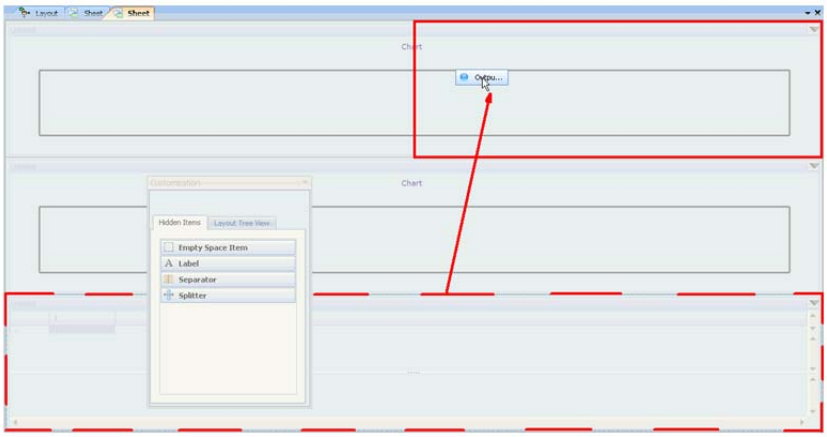


Figure 5.21 Re-arrange objects in a Sheet in 'Design Mode'

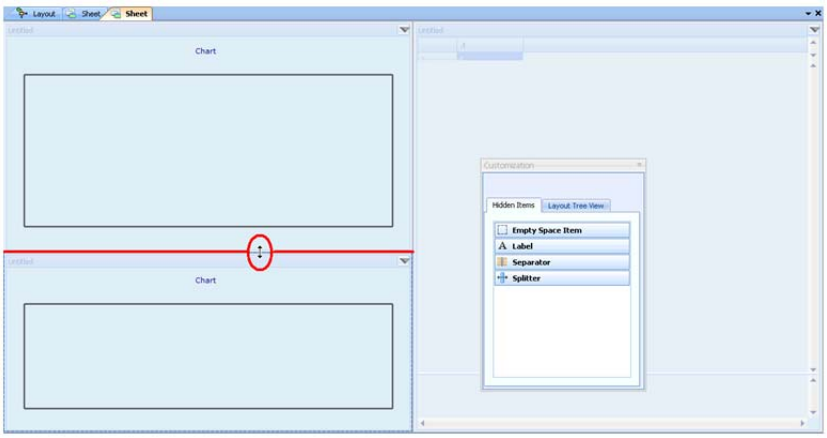


Figure 5.22 Re-size objects in a Sheet in 'Design Mode'

5.3.4 Populate Input and Output Objects Within a Sheet

Every input and output *object* is to be populated with the quantities that are to be visualised, i.e. Model Parameters and Variables respectively. Proceed as follows:

- Select the relevant Block, in the Layout Sheet or in the Model Explorer window.
- Switch to the Block Details window and further to the Parameters tab (for an input *object*) or to the Variables tab (for an output *object*).

- Select the relevant Model Parameters or Variables, drag and drop them onto the appropriate *object* (Figure 5.23). Multi-selection is supported, by holding down the “Shift” key and selecting the first and the last item of a consecutive list; or by holding down the “Ctrl” key and selecting non-consecutive items.

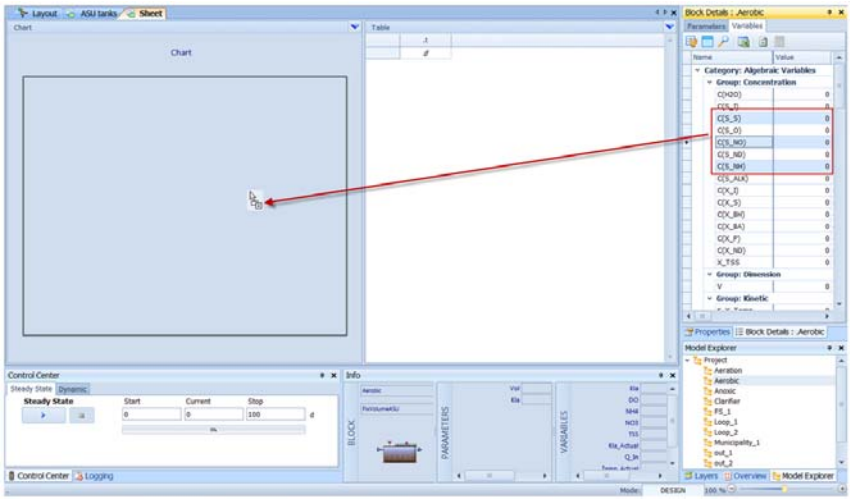


Figure 5.23 Multi-select, drag and drop variables that are to be added to a plot window

5.4 How to Manage Input Data

The Influent Tool allows for retrieving data from multiple sources (input files) and for (graphically) setting up a Fractionation Model.

Depending on the nature of the data, i.e. wastewater (defined as ASM components or as custom measurements), data or vector, the Tool comprises different steps.

5.4.1 Wastewater Input / Standard ASM vector

To access the Influent Tool, double click on a “Municipal” (“Industrial”, or “Truckload”) Block in the Layout Sheet; or select the Block and press the “Enter” key. Once the influent file for a particular input Block has been generated the first time, the corresponding Influent Sheet can be accesses through the “Sheets” button in the View | Windows.

General

- Choose “Standard”.
- “Input Category”:
  select one of the Categories available for the current Instance

- “Input Components”:
  - Add (or remove) the relevant Components from the default list.
  - Indicate which component is to be used as Solvent.
- “Output Category”:
  same as for Input Category
- Uncheck “Enable F2C conversion” if the time series is to be generated in mass units (g/d).

Data Import

- Uncheck “Synchronize” if two separate time series are going to be used for the generation of the steady-state and of the dynamic input files.
- “Simulation Input”:
  Uncheck “Interpolate” and/or “Extrapolate”, if the input file should not be interpolated and/or extrapolated.
- “Time Policy”:
  Choose between “Time Series In”, “Generate” and “Manual”; and provide appropriate settings.
- Read in one or more source files (click on the “Add” button of the “Time Series In” section) and make sure the appropriate settings (e.g. “Decimal Separator”) are set.
  In case “Time Series In” has been chosen as “Time Policy”, one of the source files needs to be used as the reference for the time scale (“Time” radio button).
- Drag the appropriate column headers from the file viewer on the right-hand side of the “Time Series In” section, to the “Time Series Out” section and make sure that all the expected “Components” be paired to a corresponding data set (“Header”). If not, establish the match manually, by opening the drop-down menu in the “Component” column.
- Make sure all the units are correct (“Unit”).
- Uncheck the “Interpolate” and “Extrapolate” options if appropriate.

Generate and Review

- Click on the “Generate” button, to process the raw data provided as source and generate the Steady-State and the Dynamic input files (*.Steady State.in.txt and *.Dynamic.in.txt respectively in the Project folder)
- The two files can be viewed and edited through the respective tab sub-pages
- Use the “Save” button to store the settings of the Influent Tool in order to be able to access it at a later stage

5.4.2 Wastewater Input / Custom vector

To access the Influent Tool, double click on a “Municipal” (“Industrial”, or “Truckload”) Block in the Layout Sheet; or select the Block and press the “Enter” key. Once the influent file for a particular input Block has been generated the first time, the corresponding Influent Sheet can be accesses through the “Sheets” button in the View | Windows

General

- Choose “Custom”.
- “Input Components”:
  - Add (or remove) the relevant Components from the default list.
  - Indicate which component is to be used as Solvent.
- “Output Category”:
  select one of the Categories available for the current Instance
- Uncheck “Enable C2F conversion” if the time series is to be generated in mass units (g/d).

Fractionation

- Connect the Input (blue) and Output (green) Blocks on the canvas (corresponding to the Custom- and to the Output Category components respectively, as indicated in the General tab page).
- Alternatively: load the default De-fractionation Layout (“Load” button in the toolbar).

Data Import

- Uncheck “Synchronize” if two separate time series are going to be used for the generation of the steady-state and of the dynamic input files.
- “Simulation Input”:
  Uncheck “Interpolate” and/or “Extrapolate”, if the input file should not be interpolated and/or extrapolated.
- “Time Policy”:
  Choose between “Time Series In”, “Generate” and “Manual”; and provide appropriate settings.
- Read in one or more source files (click on the “Add” button of the “Time Series In” section) and make sure the appropriate settings (e.g. “Decimal Separator”) are set.
  In case “Time Series In” has been chosen as “Time Policy”, one of the source files needs to be used as the reference for the time scale (“Time” radio button).
- Drag the appropriate column headers from the file viewer on the right-hand side of the “Time Series In” section, to the “Time Series Out” section and make sure that all the expected “Components” be paired to a corresponding data set (“Header”). If not, establish the match manually, by opening the drop-down menu in the “Component” column.

- Make sure all the units are correct (“Unit”).
- Uncheck the “Interpolate” and “Extrapolate” options if appropriate.

Generate and Review

- Click on the “Generate” button, to process the raw data provided as source and generate the Steady-State and the Dynamic input files (*.Steady State.in.txt and *.Dynamic.in.txt respectively in the Project folder)
- The two files can be viewed and edited through the respective tab sub-pages
- Use the “Save” button to store the settings of the Influent Tool in order to be able to access it at a later stage

5.4.3 Data Input

To access the Influent Tool, double click on a Data Input Block in the Layout Sheet; or select the Block and press the “Enter” key.

General

- Terminal:
  indicate the name of the Terminal the top-level interface variable(s) will be associated to (relevant in the scope of a coupled-model)
- Select the relevant variable from the list of top-level interface variables; or
  - Click on the “Top-level Interface Variables” button
  - Create a top-level interface variable (with “Output” as Causality attribute)
  - Select it.

Data Import

- Uncheck “Synchronize” if two separate time series are going to be used for the generation of the steady-state and of the dynamic input files.
- “Simulation Input”:
  Uncheck “Interpolate” and/or “Extrapolate”, if the input file should not be interpolated and/or extrapolated.
- “Time Policy”:
  Choose between “Time Series In”, “Generate” and “Manual”; and provide appropriate settings.
- Read in one or more source files (click on the “Add” button of the “Time Series In” section) and make sure the appropriate settings (e.g. “Decimal Separator”) are set.
  In case “Time Series In” has been chosen as “Time Policy”, one of the source files needs to be used as the reference for the time scale (“Time” radio button).

- Drag the appropriate column headers from the file viewer on the right-hand side of the “Time Series In” section, to the “Time Series Out” section and make sure that all the expected “Components” be paired to a corresponding data set (“Header”). If not, establish the match manually, by opening the drop-down menu in the “Component” column.
- Make sure all the units are correct (“Unit”).
- Uncheck the “Interpolate” and “Extrapolate” options if appropriate.

Generate and Review

- Click on the “Generate” button, to process the raw data provided as source and generate the Steady-State and the Dynamic input files (*.Steady State.in.txt and *.Dynamic.in.txt respectively in the Project folder)
- The two files can be viewed and edited through the respective tab sub-pages
- Use the “Save” button to store the settings of the Influent Tool in order to be able to access it at a later stage

5.4.4 Vector Input

To access the Influent Tool, double click on a Vector Input Block in the Layout Sheet; or select the Block and press the “Enter” key.

General

- Name:
  the name of the vector
- Terminal:
  indicate the name of the Terminal the top-level interface variable(s) will be associated to (relevant in the scope of a coupled-model)
- Category:
  select one of the Categories available for the current Instance
- Components:
  (read-only) list of components determined by the choice of the Category.

Data Import

- Uncheck “Synchronize” if two separate time series are going to be used for the generation of the steady-state and of the dynamic input files.
- “Simulation Input”:
  Uncheck “Interpolate” and/or “Extrapolate”, if the input file should not be interpolated and/or extrapolated.
- “Time Policy”:
  Choose between “Time Series In”, “Generate” and “Manual”; and provide appropriate settings.

- Read in one or more source files (click on the “Add” button of the “Time Series In” section) and make sure the appropriate settings (e.g. “Decimal Separator”) are set.
  In case “Time Series In” has been chosen as “Time Policy”, one of the source files needs to be used as the reference for the time scale (“Time” radio button).
- Drag the appropriate column headers from the file viewer on the right-hand side of the “Time Series In” section, to the “Time Series Out” section and make sure that all the expected “Components” be paired to a corresponding data set (“Header”). If not, establish the match manually, by opening the drop-down menu in the “Component” column.
- Make sure all the units are correct (“Unit”).
- Uncheck the “Interpolate” and “Extrapolate” options if appropriate.

Generate and Review

- Click on the “Generate” button, to process the raw data provided as source and generate the Steady-State and the Dynamic input files (*.Steady State.in.txt and *.Dynamic.in.txt respectively in the Project folder)
- The two files can be viewed and edited through the respective tab sub-pages
- Use the “Save” button to store the settings of the Influent Tool in order to be able to access it at a later stage

5.4.5 Create a Fractionation Model

In the “Fractionation” tab page of the Influent Tool, Input (blue) and Output (green) Blocks are displayed (corresponding to the Custom- and to the Output Category components respectively, as indicated in the “General” tab page) (Figure 5.24).

The Fractionation Model can be constructed graphically by establishing the appropriate connections (Links) between the blocks. Starting points can be the initial set of input and output blocks (as in Figure 5.24) or a pre-loaded Fractionation Layout that is to be modified.

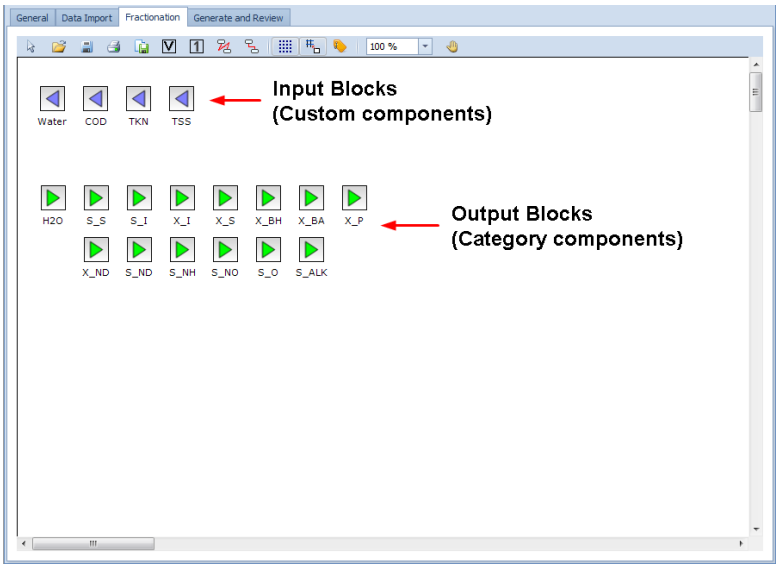


Figure 5.24 Fractionation tab page: building Blocks for the Fractionation Layout

Elements of the Fractionation Layout and Model

The building Blocks for the Fractionation Layout are “Input”, “Output”, “Variable” and “1” Blocks and can be added to the canvas by clicking the corresponding button on the toolbar (Table 5.2).

Table 5.2 Relationship between blocks of the Fractionation Layout and elements of the Fractionation Model

Block	Button	Description	Model Objects
Input	N/A	Custom component	Interface variable (input)
Constant	“1”	Constant value = 1	N/A
Parameter*	N/A	Parameter	Parameter
Variable	“1”	Calculated variable	Algebraic state variables
Output	N/A	Category component	Interface variable (output)

* Fractionation model parameters can be added by selecting the “Parameters …” item in the context menu

Place Blocks on the Layout

To place a Constant or Variable Block on the layout, click on the corresponding button in the toolbar and drop the Block on the canvas.

A Block can be renamed, a suitable description can be attached to it and its label displayed or not, through the Rename Component dialog (cf. Figure 5.25): double-click on the Block to access it.

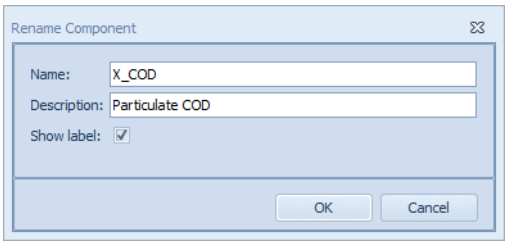


Figure 5.25 Rename Component dialog

All Blocks allows for an unlimited number of connections (incoming or outgoing, depending on the type of Block). However, mass conservation over a block is not verified.

Connect Blocks

To establish a connection between two Blocks, proceed as follows:

- select the Polygonal or the Orthogonal line on the toolbar,
- hover over the outgoing terminal of the first Block (the terminal will be *activated*, Figure 5.26),
- click on the terminal and extend the line until it reaches the ingoing terminal of the second Block,
- click to terminate the connection.

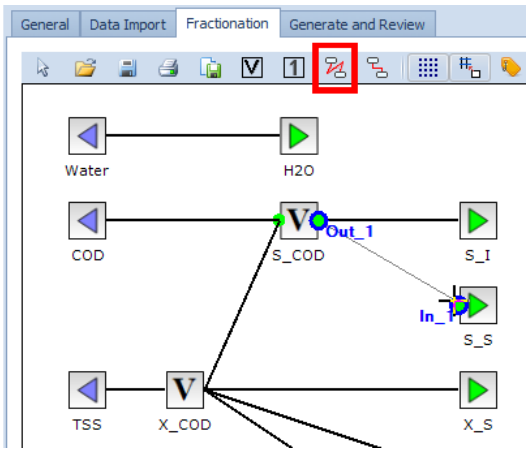


Figure 5.26 Fractionation Model: draw a Connection

In the “polyline mode” , the direction of a connection line can be changed by placing an anchor: while extending the line from the start terminal to the end terminal, click on the drawing board and change direction of movement.

Quit the connection *mode*, by clicking the “Select” button in the toolbar.

Assign a Weight to a Connection

The *equations* of the Fractionation Model are to be constructed by assigning weights to every connection of the Layout – default weight is 1.

A weight can be set by double-clicking on the connection and typing in the appropriate expression (Figure 5.27): prepend the identifier of a parameter, state- or interface variable by “parameters.”, “state.” or “interface.” respectively (usual MSL language conventions apply).

To create a Fractionation model parameter:

- Right-click in the canvas and choose “Parameters ...” from the Context menu.
- Add a new element to the table and provide the necessary properties (Name, Type, Unit, Default Value, …).

A fractionation **state variable** corresponds to a variable Block used in the Layout.

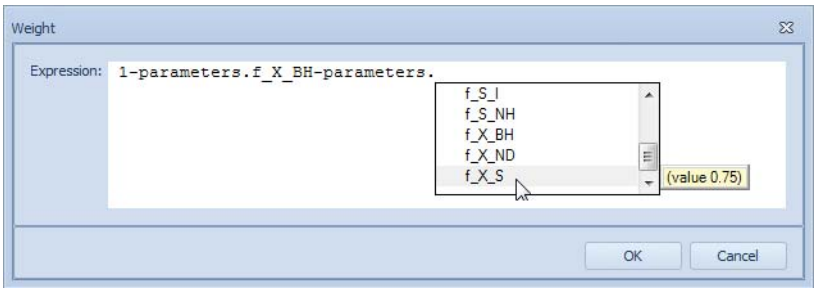


Figure 5.27 Expression editor to assign a weight to a connection

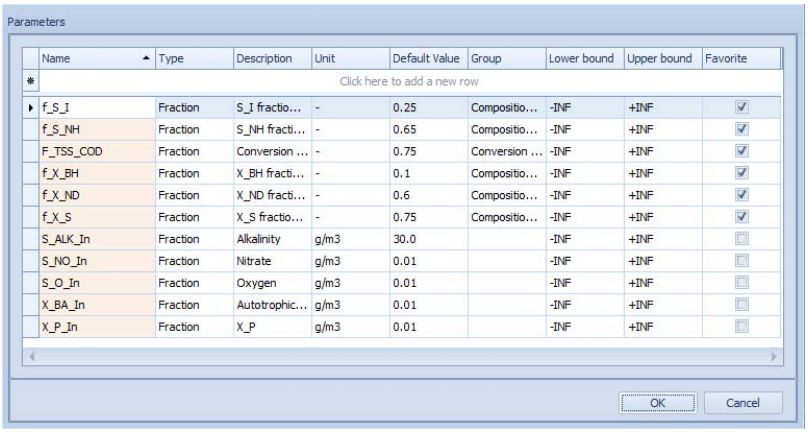


Figure 5.28 Parameters of the Fractionation model

5.5 How to Manage Simulation Inputs

The values of a Model Parameter can be changed through the Block Details pane. This, however, may be not very convenient as it entails a sequence of manual operations to locate the desired parameter and eventually edit its value.

The following means of altering the value of a Model Parameter are available in WEST:

- Through an input data set (Section 5.4.3).
- Through a control action (Section 5.2.15).
- Through an input Slider, Field, Radiobutton, Combobox or Checkbox.

The first two options are restricted to the sub-class of the Model Parameters termed Manipulated Variables (in essence, interface variables).

Sliders, Fields, Radiobuttons, Combo- and Checkboxes can act on both categories, i.e. a true Parameters and a Manipulated Variable: in the first instance, they will be disabled during the simulation and the parameter value can only be assigned prior to starting a simulation.

5.5.1 Use of a Slider

A Slider pane acts as a *container* for a number of Sliders that are to be created through drag and drop of the desired Model Parameter from the Block Details window onto the Slider pane (Section 5.3.4).

Now, a Slider can be regarded as a *manual controller*, that is: it enables to manually vary the Parameter value by sliding the thumb of the Slider.

It is typically associated to a Plot showing the effect of the change in the Parameter value on a Model Variable.

The properties that are to be set are primarily the upper (**Max**) and lower limits (**Min**) of the interval, in which the Parameter can vary by directly editing the text boxes on the Slider.

It may also be convenient to adapt the Resolution (through the **LargeChange** and **SmallChange** properties) and the distance of the tick marks (property: **Frequency**) in the Properties window of the Slider.

5.5.2 Use of an Input Field

An Input Field pane acts as a container for a number of Fields that are to be created through drag and drop of the desired Model Parameter from the Block Details window onto the Field pane (see section 5.3.4, page 175).

Now, a Field can be regarded as a *manual controller*, i.e.: it enables to manually vary the Parameter value by inputting the desired value.

It is typically associated to a Plot showing the effect of the change in the Parameter value on a Model Variable.

5.5.3 Use of a Radiobutton

A Radiobutton pane acts as a container for a number of Radiobuttons that are to be created by dragging-dropping the desired Model Parameter from the Block Details window onto the Radiobutton pane (see section 5.3.4, page 175).

Now, a Radiobutton can be regarded as a manual controller, i.e. it enables to manually vary the Parameter value by selecting the desired item.

The key property of a Radiobutton (“Items”) is the list of values of the underlying model Parameter and the corresponding labels to be displayed on the GUI. Buttons within a Radiobutton group are mutually exclusive and can be displayed in columns (“Columns” property).

Figure 5.29 exemplifies the use and the settings of a Radiobutton.

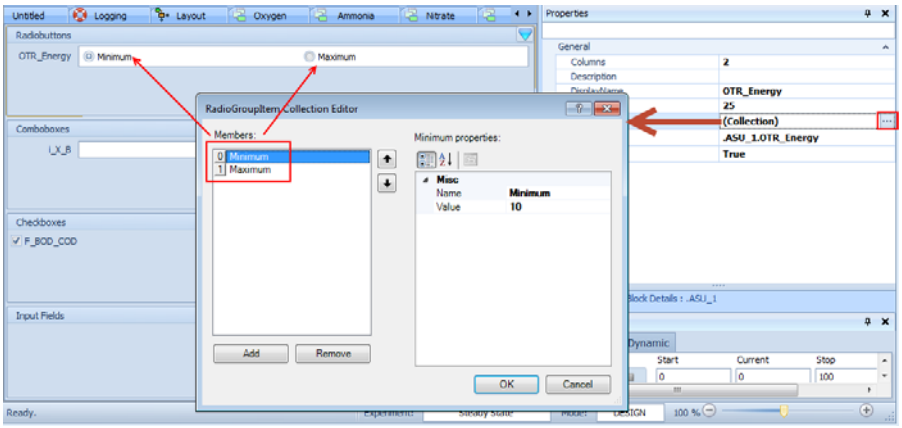


Figure 5.29 Example of use of radiobuttons

5.5.4 Use of a Combobox

A Combobox pane acts as a container for a number of Comboboxes that are to be created by dragging-dropping the desired Model Parameter from the Block Details window onto the Combobox pane (see section 5.3.4, page 175).

Now, a Combobox can be regarded as a manual controller, i.e. it enables to manually vary the Parameter value by selecting the desired item.

The key property of a Combobox (“Items”) is the list of values of the underlying model Parameter and the corresponding labels to be displayed on the GUI. The items of the drop-down list are mutually exclusive.

5.5.5 Use of a Checkbox

A Checkbox pane acts as a container for a number of Checkboxes that are to be created by dragging-dropping the desired Model Parameter from the Block Details window onto the Combobox pane (see section 5.3.4, page 175).

Now, a Checkbox can be regarded as a manual controller, i.e. it enables to manually vary the Parameter value by changing the Checkbox state, i.e. checked or unchecked, which correspond to ‘1’ or ‘0’ respectively.

5.5.6 Remove a Slider, Field, Radiobutton, Combobox or Checkbox

An input widget can be removed from its pane, through the Show Options menu (available by clicking on the top right corner ▼) by choosing the corresponding ‘Remove’ option and further selecting the Model Parameter to be removed.

5.6 How to Manage Output Data

The Effluent Tool allows for setting up (graphically) a De-fractionation Model.

Depending on the nature of the output data, i.e. wastewater (defined as ASM components or as custom measurements), data or vector, the Tool comprises different steps.

5.6.1 Wastewater Output / Standard ASM vector

To access the Effluent Tool, double click on an “Effluent” (or “Waste”) Block in the Layout Sheet; or select the Block and press the “Enter” key.

General

- Choose “Standard”.
- “Input Category”:
  select one of the Categories available for the current Instance
- “Output Category”:
  same as for Input Category
- “Output Components”:
  - Add (or remove) the relevant Components from the default list.
  - Indicate which component is to be used as Solvent.
- Uncheck “Enable F2C conversion” if the time series is to be generated in mass units (g/d).

Generate

- Click on the “Generate” button, to effectively create the (pass-through) Effluent Model.

5.6.2 Wastewater Output / Custom vector

To access the Effluent Tool, double click on an “Effluent” (or “Waste”) Block in the Layout Sheet; or select the Block and press the “Enter” key.

General

- Choose “Custom”.
- “Input Category”:
  select one of the Categories available for the current Instance
- “Output Components”:
  - Add (or remove) the relevant Components from the default list.
  - Indicate which component is to be used as Solvent.
- Uncheck “Enable F2C conversion” if the time series is to be generated in mass units (g/d).

De-fractionation

- Connect the Input (blue) and Output (green) Blocks on the canvas (corresponding to the Output Category- and to the Custom components respectively, as indicated in the General tab page).
- Alternatively: load the default De-fractionation Layout (“Load” button in the toolbar).

Generate

- Click on the “Generate” button, to effectively create the Effluent Model.

5.6.3 Data Output

To access the Effluent Tool, double click on a Data Output Block in the Layout Sheet; or select the Block and press the “Enter” key.

General

- Terminal:
  indicate the name of the Terminal the top-level interface variable(s) will be associated to (relevant in the scope of a coupled-model)
- Select the relevant variable from the list of top-level interface variables; or
  - Click on the “Top-level Interface Variables” button
  - Create a top-level interface variable (with “Output” as Causality attribute)
  - Select it.

5.6.4 Vector Output

To access the Effluent Tool, double click on a Vector Output Block in the Layout Sheet; or select the Block and press the “Enter” key.

General

- Name:
  the name of the vector
- Terminal:
  indicate the name of the Terminal the top-level interface variable(s) will be associated to (relevant in the scope of a coupled-model)
- Category:
  select one of the Categories available for the current Instance
- Components:
  (read-only) list of components determined by the choice of the Category.

5.6.5 Create a De-fractionation Model

In the “Defractionation” tab page of the Effluent Tool, Input (blue) and Output (green) Blocks are displayed (corresponding to the Output Category- and to the Custom components respectively, as indicated in the “General” tab page) (Figure 5.30).

The De-fractionation Model can be constructed graphically by establishing the appropriate connections (Links) between the blocks. Starting points can be the initial set of input and output blocks (as in Figure 5.30) or a pre-loaded De-fractionation Layout that is to be modified.

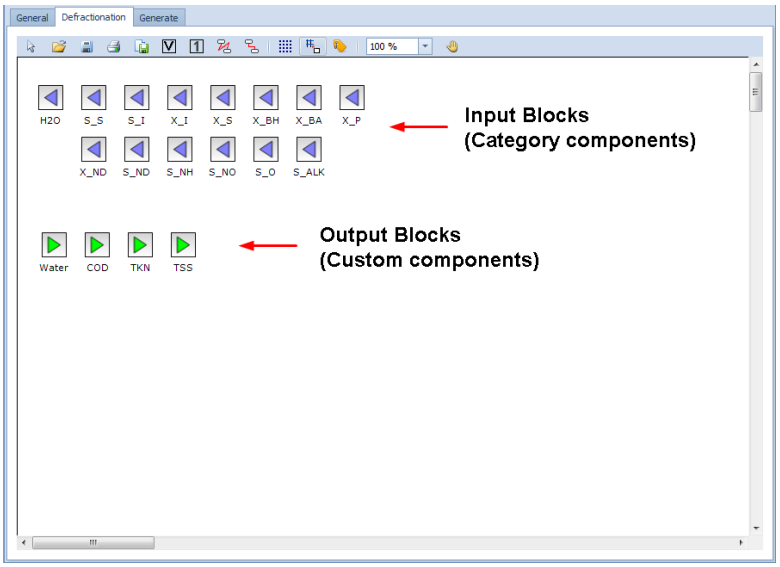


Figure 5.30 Second page of the Effluent wizard: set up the De-fractionation Model

Elements of the De-fractionation Layout and Model

The building Blocks for the De-fractionation Layout are “Input”, “Output”, “Variable” and “1” Blocks and can be added to the canvas by clicking the corresponding button on the toolbar (Table 5.3).

Table 5.3 Relationship between blocks of the De-fractionation Layout and elements of the De-fractionation Model

Block	Button	Description	Model Objects
Input	N/A	Custom component	Interface variable (input)
Constant	“1”	Constant value = 1	N/A
Parameter*	N/A	Parameter	Parameter

Table 5.3 Relationship between blocks of the De-fractionation Layout and elements of the De-fractionation Model

Block	Button	Description	Model Objects
Variable	“1”	Calculated variable	Algebraic state variables
Output	N/A	Category component	Interface variable (output)
* De-fractionation model parameters can be added by selecting the “Parameters …” item in the context menu			

Place Blocks on the Layout

To place a Constant or Variable Block on the layout, click on the corresponding button in the toolbar and drop the Block on the canvas.

A Block can be renamed, a suitable description can be attached to it and its label displayed or not, through the Rename Component dialog (Figure 5.25): double-click on the Block to access it.

All Blocks allows for an unlimited number of connections (incoming or outgoing, depending on the type of Block). However, mass conservation over a block is not verified.

Connect Blocks

To establish a connection between two Blocks, proceed as follows:

- select the Polygonal or the Orthogonal line on the toolbar,
- hover over the outgoing terminal of the first Block (the terminal will be activated, Figure 5.31),
- click on the terminal and extend the line until it reaches the ingoing terminal of the second Block,
- click to terminate the connection.

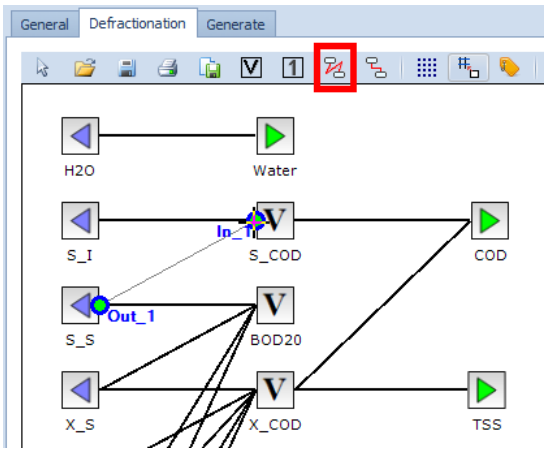


Figure 5.31 De-fractionation Model: draw a Connection

In the “polyline mode” , the direction of a connection line can be changed by placing an anchor: while extending the line from the start terminal to the end terminal, click on the drawing board and change direction of movement.

Quit the connection mode, by clicking the “Select” button in the toolbar.

Assign a Weight to a Connection

The equations of the De-fractionation Model are to be constructed by assigning weights to every connection of the Layout – default weight is 1.

A weight can be set by double-clicking on the connection and typing in the appropriate expression (Figure 5.27): prepend the identifier of a parameter, state- or interface variable by “parameters.”, “state.” or “interface.” respectively (usual MSL language conventions apply).

To create a Fractionation model parameter:

- Right-click in the canvas and choose “Parameters ...” from the Context menu.
- Add a new element to the table and provide the necessary properties (Name, Type, Unit, Default Value, …).

A de-fractionation state variable corresponds to a variable Block used in the Layout.

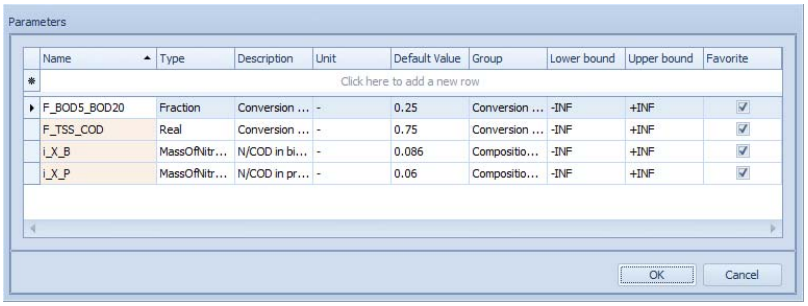


Figure 5.32 Parameters of the De-fractionation model

5.7 How to Manage Simulation Outputs

The following means of visualising simulation outputs are available in WEST:

- Output Fields
- Tables
- Plots
- Gauges

5.7.1 Use of an Output Field

An output Field pane acts as a *container* for a number of Fields which are to be created by dropping the desired Model Variable from the Block Details window onto the Field pane (Section 5.3.4).

As the simulation progresses, the **current value** of the Model State Variable associated with the Field will be displayed.

The elements that are to be set are primarily the **Communication Interval** (i.e. the frequency at which the simulation output is sent to the Field) and the **Type of Interpolation** (i.e. whether output quantity values are interpolated), by default set to 0.01 day and Linear respectively.

All the settings of an output Field can be accessed from the “Output tab page” of the Simulation Properties window (Section 4.1.1).

5.7.2 Use of a Table

A Table can display a number of columns that are to be associated with a Model Variable by dragging the desired Variable from the Block Details window onto the Table pane (Section 5.3.4).

As the simulation progresses, the **current value** of the Model State Variable associated with each column of the Table is dropped into the Table at pre-defined time intervals.

The elements that are to be set are primarily the **Communication Interval** (i.e. the frequency at which the simulation output is sent to the Table) and the **Type of Interpolation** (i.e. whether output quantity values are interpolated), by default set to 0.1 day and Linear respectively.

All the settings of a Table can be accessed from the “Output tab page” of the Simulation Properties window (Section 4.1.1).

5.7.3 Use of a Plot

The following five categories of plots are available:

- **Time series**
  This is the default plot type that can be used for a standard time series (i.e. Model Variable).
  Data series can be added through drag and drop from the Block Details pane. Sub-categories are: line, fast line, point and column (Figure 5.33).

- **Current value**
  This plot type shows the current value (at every time step) of a standard time series (i.e. Model Variable).
  Data series can be added through drag and drop from the Block Details pane. Sub-categories are: line, fast line, point, column, bar, pie, area, surface and color grid (Figure 5.34).

- **Y(X)**
  This plot type can be used for a generic variable (not necessarily a time series).
  Data series can be added through drag and drop from the Block Details pane (in case of time series); or through the Add Series item of the plot sub-menu (accessible by clicking on the blue arrow on the top right corner). Sub-categories are: line, fast line, point and column (Figure 5.35).

- **Z(X,Y)**
  This plot type shows the relationship between 3 variables.
  Data series can be added through the Add Series item of the plot sub-menu (accessible by clicking on the blue arrow on the top right corner). Sub-categories are: surface, point 3D column 3D and color grid (Figure 5.36).

- **Histogram**
  This plot type is to be used for a not time-dependent variable: data series can be added through the Add Series item of the plot sub-menu (accessible by clicking on the blue arrow on the top right corner). Sub-categories are: vertical and horizontal (Figure 5.37).
  Additional settings need to be provided to create the plot:
  - Automatic Minimum, or specify the Minimum
  - Automatic Maximum, or specify the Maximum
  - Number of Classes or Interval
  - Cumulative

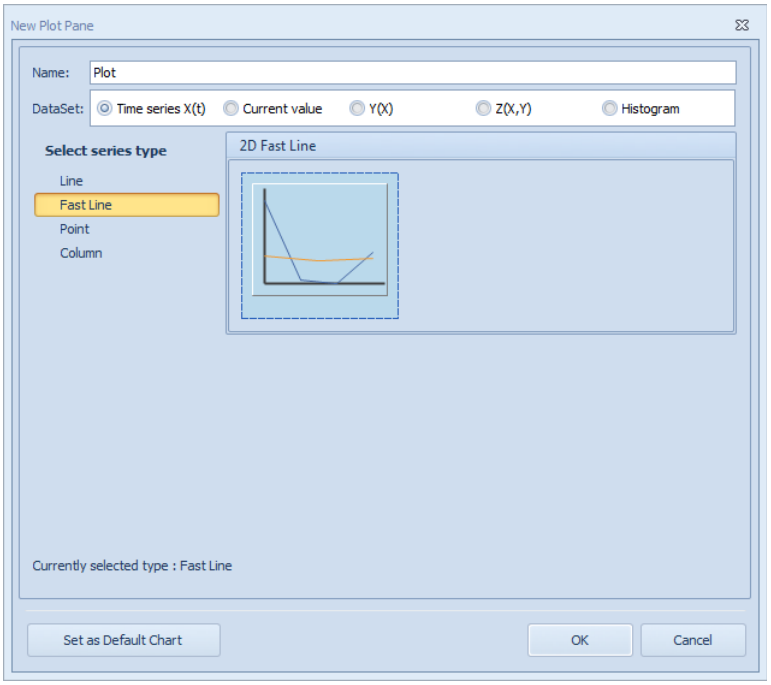


Figure 5.33 Plot options for the Time series data set

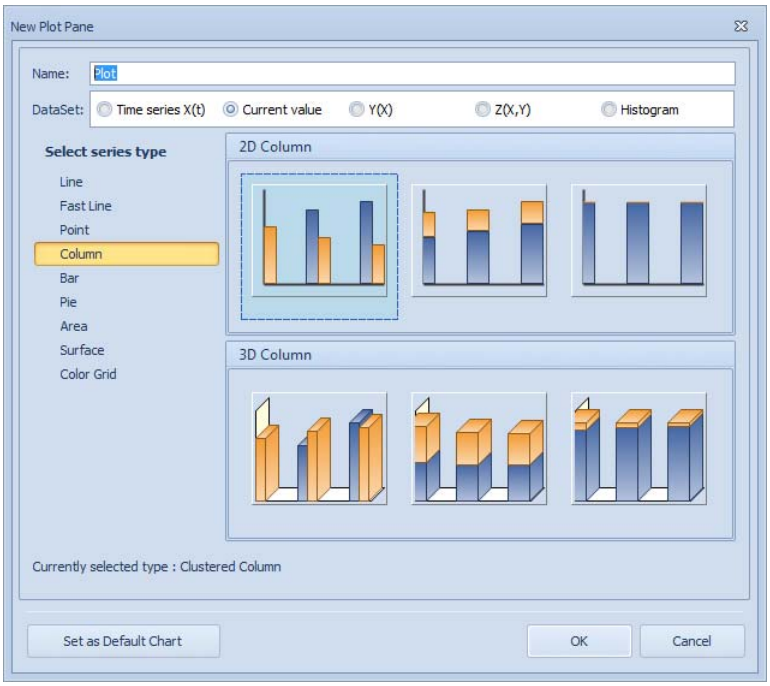


Figure 5.34 Plot options for the Current value data set

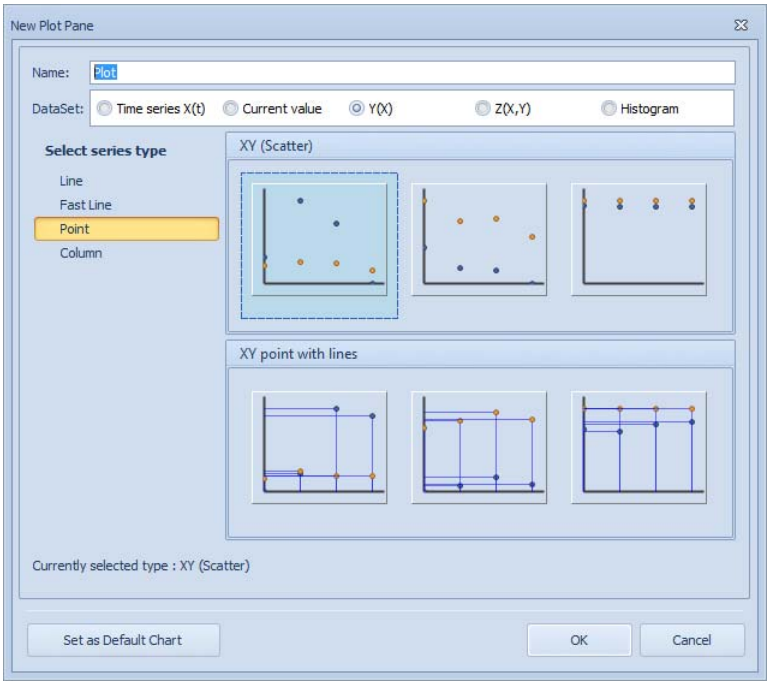


Figure 5.35 Plot options for the Y(X) data set

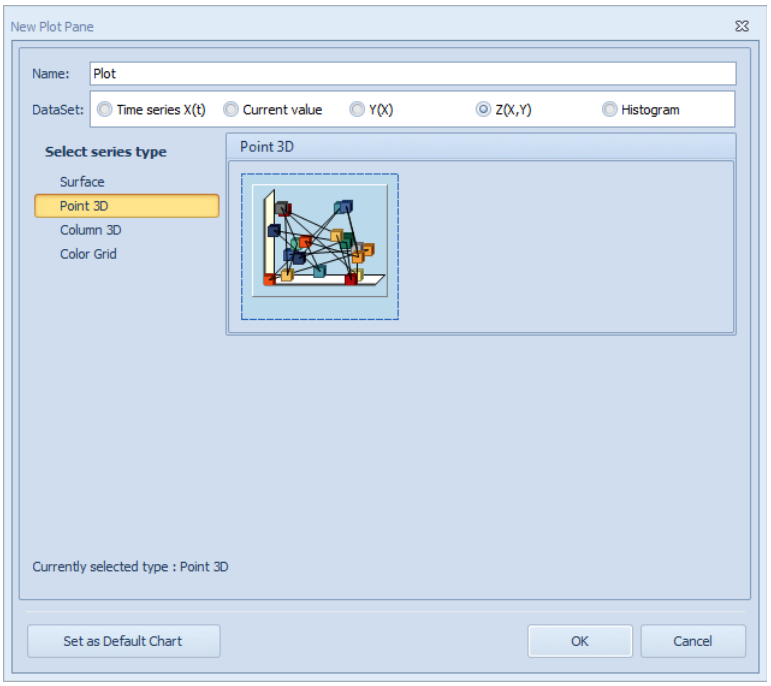


Figure 5.36 Plot options for the Z(X,Y) data set

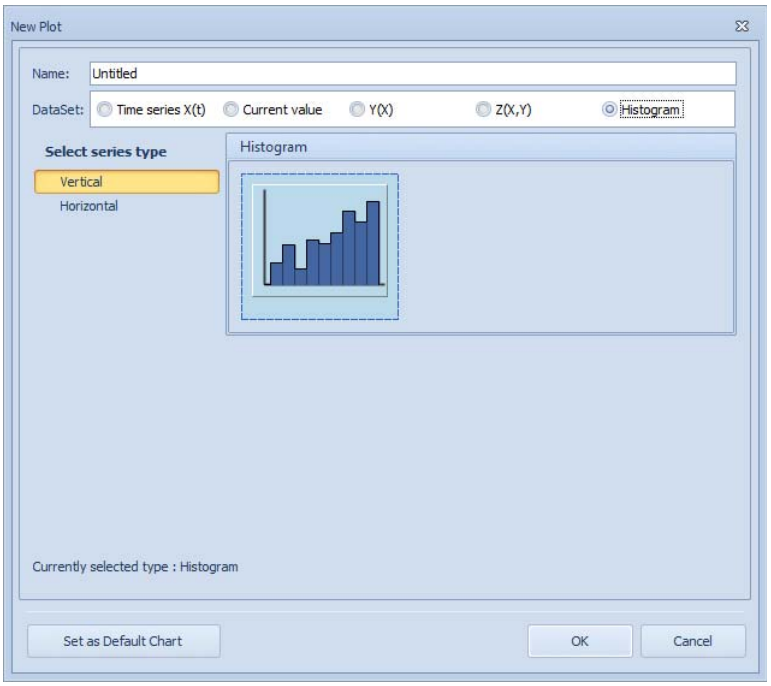


Figure 5.37 Plot options for the Histogram data set

A Plot can display two kinds of data series:

- **Model Variables**
  These objects are defined at the level of each (sub)model and therefore available in the corresponding Block Details pane. Model Variables can be dragged and dropped to a Plot (Section 5.3.4)

- **Objective- and Experiment-specific Variables**
  These quantities are computed at the end of (each) simulation or at the end of the experiment and are therefore not associated to any particular (sub)model, hence not available through the Block Details.
  They can therefore only be added to a Plot via the “Add Series” item of the plot menu (accessible by clicking on the blue arrow on the top right corner of the Plot).

To add a data series to a Plot, via the “Add Series” dialog window provide the following information.

- **Name**
  Name of the data series (as it appears e.g. in the Legend). If not provided, the data series will be assigned the fully qualified name of the selected variable.

- **Source Type**
  Type of source for the dataset: File or Experiment (i.e. the WEST simulation engine).

- **Location** (only if Source Type is set to ‘File’)
  File that contains the datasets for the X- and Y- (and Z-, if 3D) items.

- **Source Name** (only if Source Type is set to ‘Experiment’)
  Name of the data source. Options are as follows:
  - *Simul* (for all experiment types)
    Simulation experiment. Not available for a Histogram type plot.
  - *Sens* (for LSA)
    Sensitivity evaluation
  - *SensFunct*(<VarName>)(<ParamName>) (for LSA)
    Sensitivity function for the <ParamName> / <VarName> pair
  - *SimulRef* (for LSA)
    Reference simulation
  - *Simul(Backward)(<ParamName>)* (for LSA)
    Simulation with backward perturbation of <ParamName>
  - *Simul(Forward)(<ParamName>)* (for LSA)
    Simulation with forward perturbation of <ParamName>
  - *MC* (for GSA and UA)
    Monte Carlo evaluation
  - *Optim* (for PE)
    Optimization evaluation

- *Scen* (for SA)
  Scenario Analysis evaluation
- *Stats* (for UA)
  Statistical evaluation

- **Description**
  (optional) Description of the data series.

- **Quantities**
  Quantities that are to be plotted on the chart. Depending on the type of plot, the layout of this area may be different.

  For a conventional Line Plot (Time series, Y(X) or Z(X,Y): cf. Figure 5.38), the data series is to be provided through the following items:

  - *X-item* (data points for the horizontal axis)
  - *Y-item* (data points for the vertical axis)
  - *Z-item* (data points for the depth axis – for a 3D chart)

  For a Bar Chart (Current Value: cf. Figure 5.39), multiple data series can be added to the plot by adding items to the list of Quantities.

  For a Histogram plot (cf. Figure 5.40), a single data series is to be added through the single drop-down; and additional settings are to be provided with respect to the Minimum and Maximum of the interval, to the Number of Classes or to the width of every class and to whether values are to be cumulated.

  The quantities can be all model parameters and variables, if the Source Name is ‘Simul’; or ‘RunNo’ and other experiment-specific variables (e.g. objectives), for every other Source Name.

The elements that are to be set are primarily the **Communication Interval** (i.e. the frequency at which the simulation output is sent to the Plot) and the **Type of Interpolation** (i.e. whether output quantity values are interpolated), by default set to 0.01 day and Linear respectively.

All the settings of a Plot can be accessed through the Properties window, by clicking the blue triangle and select ‘Plot Properties’ or one of the plot series properties.

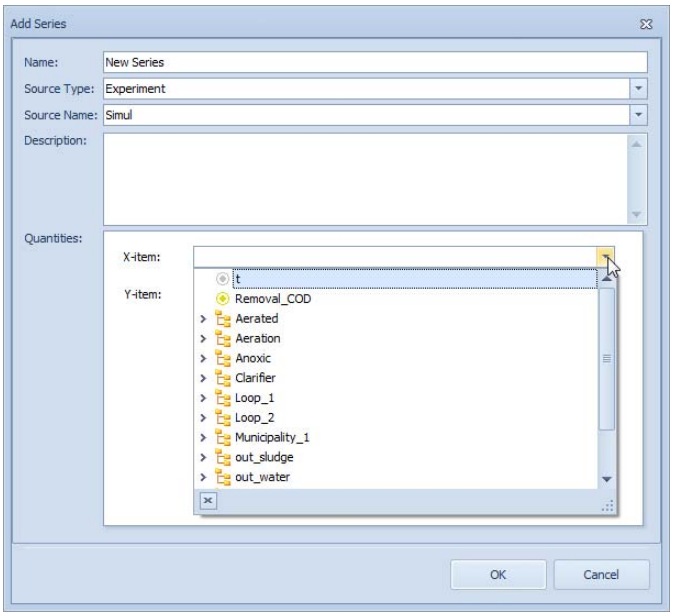


Figure 5.38 Add Series dialog window for a Line type plot

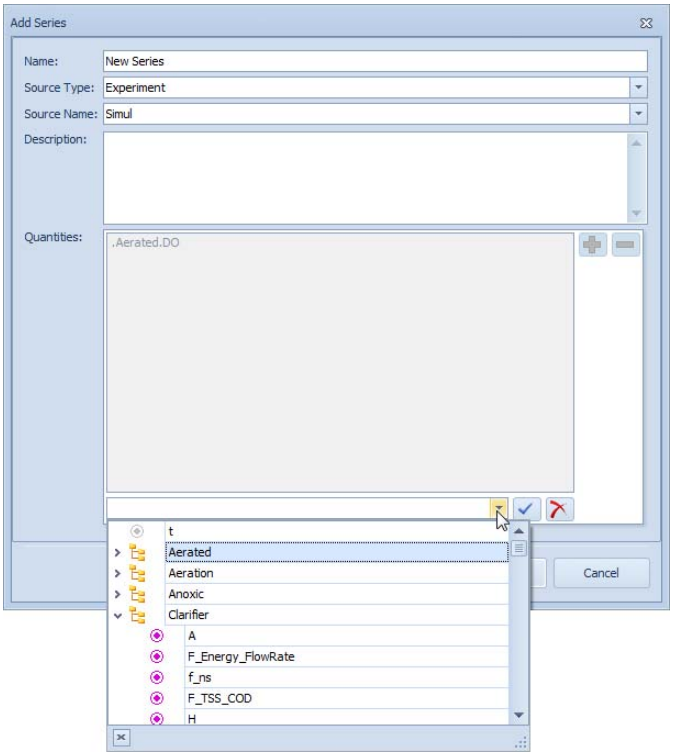


Figure 5.39 Add Series dialog window for a Bar chart

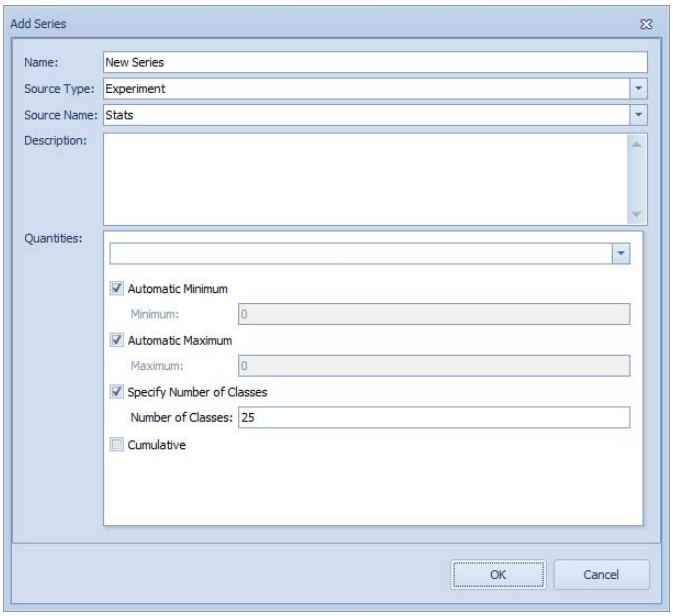


Figure 5.40 Add Series dialog window for a Histogram plot

5.7.4 Use of a Gauge

There are different types of Gauges possible, based on its shape:

- Circular Full
- Circular Half
- Circular Quarter
- Circular Three Fourth
- Circular Wide

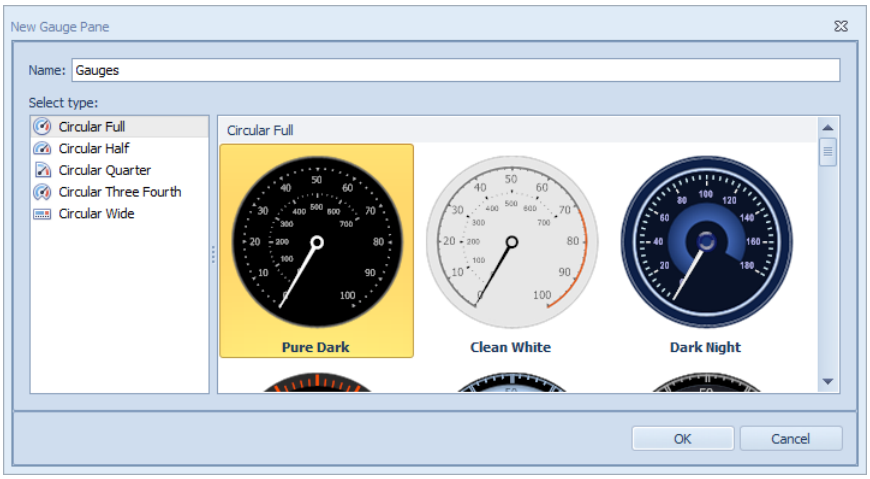


Figure 5.41 Creating a Gauge

A Gauge can display two kinds of data:

- **Model Variables**
  These objects are defined at the level of each (sub)model and therefore available in the corresponding Block Details pane. Model Variables can be dragged and dropped to a Gauge (Section 5.3.4).

- **Objective- and Experiment-specific Variables**
  These quantities are computed at the end of (each) simulation or at the end of the experiment and are therefore not associated to any particular (sub)model, hence not available through the Block Details.
  They can therefore only be added to a Gauge via the “Add Gauge” item of the Gauge menu (accessible by clicking on the blue arrow on the top right corner of the Gauge).

To add a Gauge to a Gauge pane, via the “Add Gauge” dialog window provide the following information:

- **Name**
  Name of the Gauge

- **Source Type**
  Type of source for the dataset: File or Experiment (i.e. the WEST simulation engine).

- **Location** (only if Source Type is set to ‘File’)
  File that contains the datasets for the X- and Y- (and Z-, if 3D) items.

- **Source Name** (only if Source Type is set to 'Experiment')
  Name of the data source.
  Options are as follows:
  - Simul (for all experiment types): simulation experiment
  - Sens (for LSA): sensitivity evaluation
  - SensFunct(<VarName>)(<ParamName>) (for LSA): sensitivity function for the <ParamName> / <VarName> pair
  - SimulRef (for LSA): reference simulation
  - Simul(Backward)(<ParamName>) (for LSA): simulation with backward perturbation of <ParamName>
  - Simul(Forward)(<ParamName>) (for LSA): simulation with forward perturbation of <ParamName>
  - MC (for GSA and UA): Monte Carlo evaluation
  - Optim (for PE): optimization evaluation
  - Scen (for SA): scenario Analysis evaluation
  - Stats (for UA): statistical evaluation

- **Description**:
  (optional) Description of Gauge.

- **Quantity**:
  Model quantity assigned to the Gauge.

The elements that are to be set are primarily the Communication Interval (i.e. the frequency at which the simulation output is sent to the Gauge) and the Type of Interpolation (i.e. whether output quantity values are interpolated), by default set to 0.01 day and Linear respectively.

All the settings of a Gauge can be accessed through the Properties window, by the blue triangle and select 'Gauge Properties Pane' or one of the individual Gauge Properties.

5.7.5 Remove Output Field or Data Series From a Table, a Plot or a Gauge

An Output Field can be removed from the Field pane that it belongs to, by choosing the "Remove Field" option in the sub-menu (available by clicking on the top right corner ▾) and further selecting the Model Variable to be removed.

Similarly can a specific Column be removed from a Table; a data Series from a Plot; or a Gauge from a Gauge pane.

5.7.6 Output Data to File

Another non-graphical output channel (thus not accessible directly through the Insert | Output menu) is the output to a text file.

This channel is disabled by default.

To create an output file, proceed as follows:

- Click on the “Simulation” button in the Project | Properties menu.
- Select the “Output” tab page and the “File” channel from the drop-down (cf. Figure 5.42).
- Enable the channel.
- Make sure the Communication Interval is greater than 0 (i.e. all data points) to avoid ending up with very large output files and possibly slowing down the simulation
- Add the relevant data series:
  - Click on the “Add” button next to the “Quantities” box
  - Select the desired quantity in the project treeview (as show in the figure)
  - Confirm the selection

Remember that I/O settings are experiment-specific, that is: they apply to the experiment that is currently active. The current experiment is the one selected in the Control Centre pane – and is visible on the right side of the status bar.

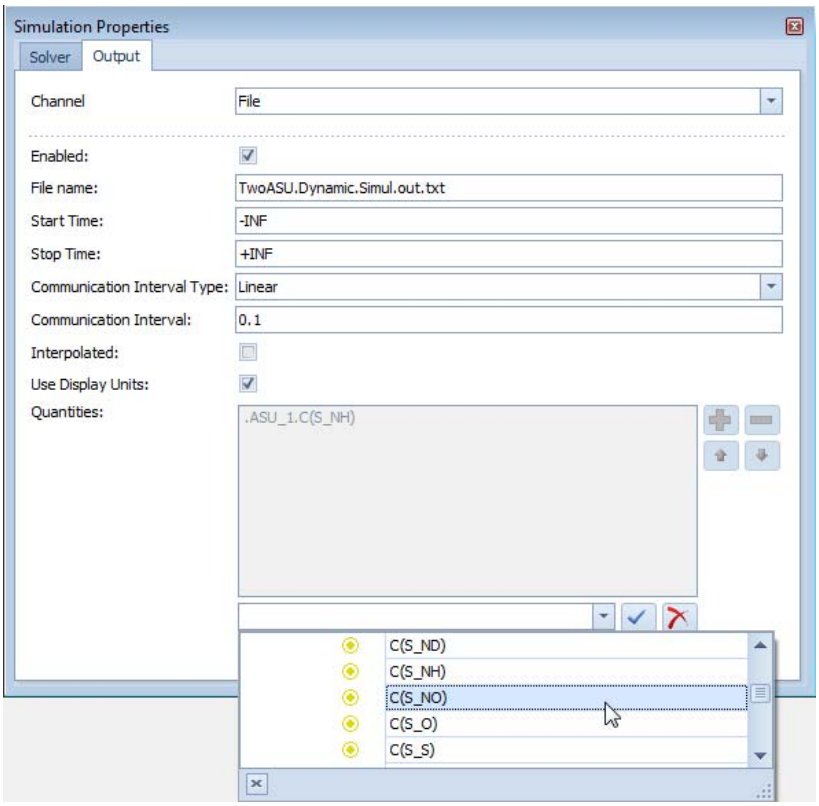


Figure 5.42 Set up a file Output for a Simulation

5.7.7 Export Results to Excel

The current State of a sub-model, i.e. the values of the Model Parameters and the current values of the Manipulated and State Variables, can be transferred to an Excel spreadsheet.

In essence, this is a snapshot of the state of a particular Block.

Proceed as follows:

- Select the Block of interest on the Layout Sheet or in the Model Explorer.
- Open the Block Details window (that will display the Properties of the Block).
- In the toolbar of the Parameters (or Variables) tab page, click on the “Save” button and choose Excel.

5.8 How to Run Dynamic Simulations

Once the Plant Layout has been set up in the Layout Sheet (Section 5.2) and the underlying Model has been generated, the Influent File, the Influent Model and the Effluent Model have been generated (Sections 5.4 and 5.6); and the relevant input and output objects have been created to enable interacting with the simulation; there is a crucial stage to undertake before the experiment eventually can be run, i.e. the initialisation of the model.

5.8.1 Initialise the Model(s)

Prior to executing the simulation, the Plant Model and each of its sub-models are to be initialised, that is: the Model Parameters are to be assigned an appropriate value and the Derived State Variables are to be given meaningful or realistic **initial values**.

In first approximation, most of the default values that are pre-set for the Model Parameters will be acceptable; the user can only set the **process-specific parameters**, typically the size of the units, the flow rates, etc.

The initialisation of Derived State Variables is a precondition for the solution of any system of differential equations; and this is the case of the dynamic models in WEST. Initialising the system means assigning *correct* initial values to the Derived State Variables of the system, which in turn implies that the system of differential equations be solved at time zero.

This is evidently impractical if not impossible. A work-around is the following:

- Assign non-zero initial values to the Derived State Variables: this is sufficient to avoid divisions by zero that would result in 'NAN' values as output.
- Run a dynamic simulation, under constant influent conditions for a sufficiently long period of time so that the initial (arbitrary) conditions are no longer relevant: the system will reach its steady-state that is all variables will show negligible change over time.
- The *end-state* of the system, i.e. its steady-state, is now a *realistic* (no longer arbitrary) initial condition for the dynamic simulation.
- Re-run the dynamic simulation, using the end-state as *initial-state* and the dynamic influent file.

In WEST, this is the default 'scenario' and therefore:

- for most models included in the WEST model library, the Derived State Variables are initialised to non-zero values;
- the Influent Tool produces a steady-state influent file by averaging the dynamic influent file;

- a dynamic simulation is always coupled to a steady-state simulation: this latter is run first and the end-state is automatically copied into the initial state for the subsequent dynamic simulation (unless the corresponding option is turned off in the Popular page of the WEST Option dialog).

This work around may fail in some special situations, i.e. the non-zero initial condition and long simulation are not sufficient for the system to converge.

In such circumstances, the only way is to carefully initialise the system offline and providing good approximations of initial values to the Derived State Variables.

5.8.2 Execute the Experiment

The *base* experiment type in WEST is the **Objective Evaluation** experiment (Section 4) which is comprised of two individual experiments, i.e. a Steady-State and a Dynamic Simulation, each relying on the corresponding input file that is produced by the Influent Tool (one pair of files is generated for each Input Block of type Wastewater).

The two simulations can be run independently, through the Control Center window, although if a Dynamic simulation is executed without a Steady-State being run up front, a warning message is issued.

Once a Steady-State simulation is completed, the Derived State Variables are reset to the end-state that the system has reached, i.e. the model re-initialised.

To start, pause and stop the simulation, press the “Start” , “Pause” or “Stop” button, respectively.

Once a simulation is paused, it can be resumed by pressing the “Start” (Continue) button, whereas if it has been stopped it can only be re-started from time zero.

5.9 How to Create a Report

Various elements of a Project can be gathered to create a standard Report.

These elements are to be selected through the treeview shown in Figure 5.43 and are grouped in the following categories – corresponding to the predefined headings of the Report:

- **General Information**:
  The textual description of the Project (“General” button of the Project | Properties menu).

- **Notes**:
  If one or more Notes objects are present, their content can be embedded in the Report.
- **Layout**:
  The content of the Layout Sheet (as image).
- **Sheets**:
  For every Sheet in the Project, individual items can be selected and added to the Report – the content of a Table will be editable.
- **Input files**:
  The content of the selected file(s) will be incorporated in the Report in form of a table.
- **Output files**:
  As for Input files.
- **Model Quantities**:
  The *current* value of Parameters and Variables for individual sub-models and/or pertaining to one of these general groups:
  - Parameters
  - Input Variables
  - Output Variables
  - Algebraic Variables
  - Derived Variables
  - Favorites:
    the list of favourite parameters and variables shown in the Block Summary of every Block.

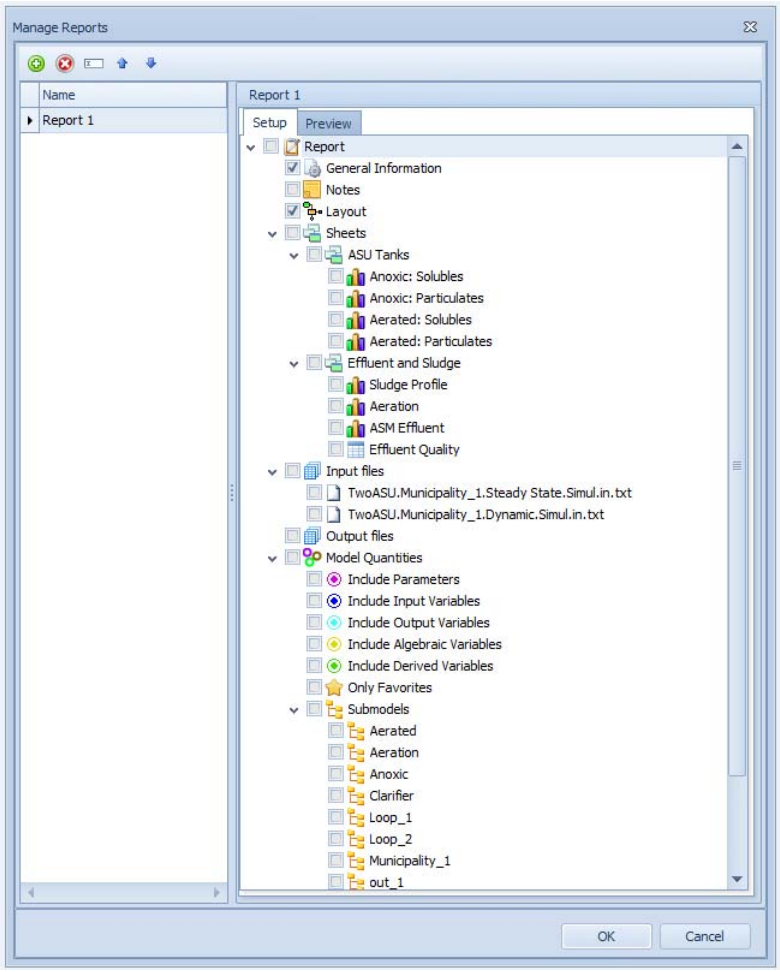


Figure 5.43 Manage Reports dialog

To create a Report, proceed as follows:

- Click on the “Reports” button of the Project | Miscellaneous menu.
- A first (empty) report object is created by default and named “Report1”.
  - Reports can be added (and removed), renamed and sorted, using the toolbar of the of the dialog window.
- Select the relevant elements from the treeview.
- (Optional) preview the Report in the Preview tab page.
  - The generation of a Report can be a lengthy process: a progress bar (and “Abort” button) are shown until the process is completed.
  - The Report can be saved in a variety of formats and printed, using the toolbar of the Preview tab page.
    By choosing the .rtf or .docx formats one can further manipulate the layout of the document in Word.

- Click on the “OK” button to confirm and close the dialog: the Report will be stored with the Project

6 Technical Specifications

For more details, please refer to the WEST Release Notes.

6.1 Prerequisites

- .NET Framework 4.0 (full profile)

6.2 How to use the VC++ compiler with WEST 32-bit

- Download Visual Studio Community from https://visualstudio.microsoft.com/downloads/
- Perform a custom installation with the "Desktop development with C++" package activated (in the Installation Details, activate all Optional features available) to ensure installation of build tools and Windows SDK
- Modify the automatically generated batch file WESTStart.bat located in …\Public\Documents\DHI\WEST\bin:
  - Set TORNADO_CC_PLATFORM to win32-msvc
  - Add the following statement as a first line to the batch files (make sure not to forget the double quotes):

    CALL "C:\Program Files (x86)\Microsoft Visual Studio\2019\Community\VC\Auxiliary\Build\vcvars32.bat"

    The path to vcvars32.bat file may differ depending on the version of Visual Studio installed (in this case, it refers to 2019 version)
    Mind the use of " " quotes, not of “ ” quotes.

  - If necessary, replace the line

    SET LIB=%LIB%;C:\Program Files (x86)\DHI\WEST\2020\lib\win32-msvc

    with the line

    SET LIB=%LIB%;C:\Program Files (x86)\DHI\WEST\2020\lib

- Save WESTStart.bat and start WEST

6.3 How to use WEST 2020 64-bit with Visual Studio Community

- Download Visual Studio Community from https://visualstudio.microsoft.com/downloads/

- Perform a custom installation with the "Desktop development with C++" package activated (in the Installation Details, activate all Optional features available) to ensure installation of build tools and Windows SDK
- Install WEST 2020 64-bit
- Modify the automatically generated batch file WESTStart.bat located in …\Public\Documents\DHI\WEST\bin using a text editor:
  - Add the following (or similar) text at the top of the .bat file:

    CALL "C:\Program Files (x86)\Microsoft Visual Studio\2019\Community\VC\Auxiliary\Build\vcvars64.bat"

    The path to vcvars64.bat file may differ depending on the version of Visual Studio installed (in this case, it refers to 2019 version).
    Mind the use of CALL and of " " quotes, not of “ ” quotes.

  - If necessary, replace the line

    SET LIB=%LIB%;C:\Program Files (x86)\DHI\WEST\2020\lib\win64-msvc

    with the line

    SET LIB=%LIB%;C:\Program Files (x86)\DHI\WEST\2020\lib

  - Optionally, if faster compilation at the expense of slower simulation is desired, add the following to WESTStart.bat:

    SET TORNADO_CC_OPTIMIZE=no

- Save WESTStart.bat and start WEST

WEST TOOLS USER GUIDE

7 Block Editor

7.1 Description

The Block Editor is an application that allows for creating and managing icon and palette libraries for use with the WEST software. An icon library is a collection of icons that can be used to represent blocks in a WEST layout. A palette library is a collection of palettes, where each palette contains references to a set of icons.

The Block Editor allows for opening at most one icon library and one palette library simultaneously at any point in time. In the case of a palette library, palettes can be created, renamed and removed as a whole. Also, icons can be added and removed from palettes. For icon libraries, icons can be added, edited and removed.

Icon editing is the most convoluted functionality that is offered by the Block Editor. It works on the basis of a canvas on which graphical shapes can be placed.

The following graphical shape types are available:

- Lines
- Polylines
- Rectangles
- Rounded rectangles
- Polygons
- Ellipses
- Curves
- Closed Curves
- Splines
- Bezier Curves
- Freehand Lines
- Text
- Rich Text
- Images

Next to placing graphical shapes on the canvas, it is also possible to identify a number of zones as terminals, by placing special terminal shapes onto the canvas.

For terminals, as well as for the icon as a whole, a number of attributes can be specified that are of relevance to the interaction between the icon and the WEST layout editor.

Icon attributes are the following:

- **Description**:
  Optional textual description
- **DisplayName**:
  Name that will appear under the icon in the Block Library
- **IconName**:
  Name that should be used to refer to the icon from model code.
- **Name**:
  Name of the icon
- **ShowWizard**:
  The type of wizard that is to be popped up by WEST when double-clicking the icon. The options are:
  - None
  - WwtpInput
  - DataInput
  - WwtpOutput
  - DataOutput

Terminal attributes are the following:

- **Name**:
  Name of the terminal
- **Causality**:
  Causality of the terminal. The options are:
  - Generic
  - Input
  - Output
- **MaximumConnections**:
  Maximum number of connections that the terminal can handle
- **Type**:
  Type of the connections that the terminal can handle. The options are:
  - Undefined
  - Generic
  - Data
  - WaterLine
  - SludgeLine
  - GasLine

7.2 Quick User Guide

1. After starting the application, the Application Menu Button can be used to create a new icon and/or palette library, or to open and activate an already existing icon and/or palette library.
2. The Add Palette Button of the Palette Library group can be used to add a new palette to the active palette library.
3. The Remove Palette Button of the Palette Library group can be used to remove the selected palette from the active palette library.
4. The Rename Palette Button of the Palette Library group can be used to rename a palette within the active palette library.
5. The Remove Block Button of the Palette Library group can be used to remove the selected icon from the active palette library.
6. An icon can be added to the active palette library through drag-and-drop from the active icon library. It is possible to add icons from different icon libraries to the same palette library.
7. By double-clicking an icon in the active palette library, the icon library that it physically resides in will be loaded and activated (if it is not already).
8. The Add Block Button of the Icon Library group can be used to add a new icon to the active icon library.
9. The Remove Block Button of the Icon Library group can be used to remove the selected icon from the active Icon Library.
10. By double-clicking an icon in the active icon library, it will be opened for editing in a separate window.
11. The icon editing process is straightforward and works in a way similar to any other drawing package.
    - The desired shape type can be selected in the Shapes group, and the shape itself can be placed on the canvas with a number of mouse clicks.
    - A shape on the canvas can be removed by clicking it and pressing the Delete Key.
    - A shape can be moved by dragging it as a whole to the new location. It can be resized by clicking it, and manipulating one of the handlers on the shape's edges. Also, it can be rotated by manipulating the rotation handler.
    - The Draw Terminal Button places a terminal on the canvas, which can be relocated to the desired position.
    - The Line, Fill and Shadow Style Button in the Style group allow for configuring the corresponding styles.
    - The buttons in the Arrange group can be used group/ungroup shapes, and to determine the drawing order (bring to front, send to back).

- The Rotate and Flip group buttons allow for rotating a shape over a number of frequently occurring angles, and to flip the shape horizontally and/or vertically.
- The Align group buttons allow for aligning all the shapes in a selected set in a number of ways.

8 Data Editor

8.1 Description

The tool allows for opening one or more Tornado data files and performing a number of typical actions such as:

- Insert/Delete Row/Column
- Change header
- Copy / paste data (e.g. from Excel)
- Rescale (x Factor + Offset) selected cells

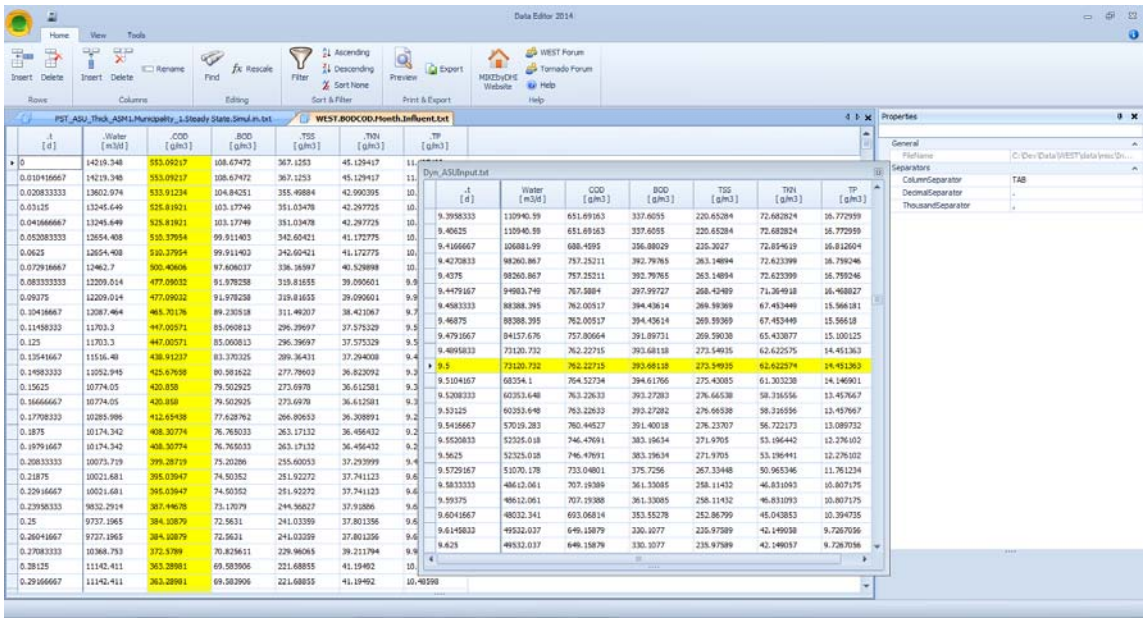


Figure 8.1 Three data files opened in the Data Editor

8.2 Quick User Guide

- In the Application menu, choose “Open” and select a valid Tornado data file (*.txt and *.xls files are allowed).
  - The content of the file will be displayed as a Table in a new Sheet (cf. Figure 8.1)
  - Multiple data files can be managed in parallel
  - The default location can be changed through Options | Advanced
- Data can be edited in various ways:

  - Rows (Columns) can be inserted or deleted
  - The column header and unit can be edited by clicking on the “Rename” button in the Home | Columns menu
  - Data (a single point, a row or a column) can be rescaled (Home | Editing menu), using a Factor and an Offset
  - Data can be sorted and filtered

- The minimum, maximum, mean and standard deviation per column can be computed by clicking on the “Summary” button in the View | File menu.
- The data file can be exported (as pdf) or saved (as text file or Excel spreadsheet).

9 Designer

9.1 Description

The tool allows for creating new design experiments based on one of the predefined templates (or opening existing design experiments) and for launching a (steady-state) simulation that, based on design criteria (e.g. influent specifications, target effluent quality, ..), yields the design quantities (e.g. volumes of the tanks, recirculation flow rates, ..) to be used to design a wastewater treatment plant.

9.2 Quick User Guide

- In the Application menu, choose “New” and select one of the available templates (Figure 9.1):
  - MLE or A2O layout
  - ATV or Metcalf & Eddy design protocol
- The corresponding Experiment will be created and the Parameters and Variables displayed in the Block Details pane
- Input the relevant Parameters
- Click on the “Simulate” button in the Home | Executor menu to execute the simulation
- The Variables tab page of the Block Details pane will get updated with the outcome of the simulation
- The same Input/Output objects as in WEST can be used to provide inputs (Sliders and Fields) and to display outputs (Plots, Tables and Fields)
- Notes (Insert | Miscellaneous menu) can be added to document the project

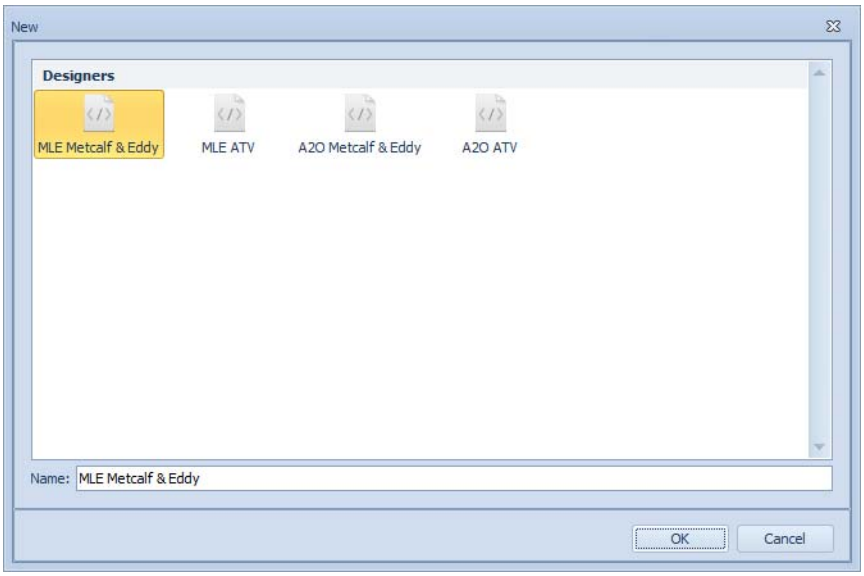


Figure 9.1 Create New design experiment

10 Model Editor

The WEST Model Editor was completely rewritten in release 2016.

Besides the completely redesigned GUI, the most notable improvements are:

- full management of the BlockLib.xml file, via the Block Library Explorer and Properties panes
- double view over the model library, i.e. file- and inclusion hierarchy trees (Files and Includes tabs of the Model Library Explorer pane)
- support for simultaneous view of multiple Gujer matrices (Conversion Models)
- support for multiple Conversion Models per Category
- support for multiple Category References per Instance
- deep Instance-based code check

This Guide presents a general illustration of the Model Editor's GUI and provides examples on how-to perform the most common operations.

A set of specialised videos will also be published on DHI's website covering a variety of more advanced operations.

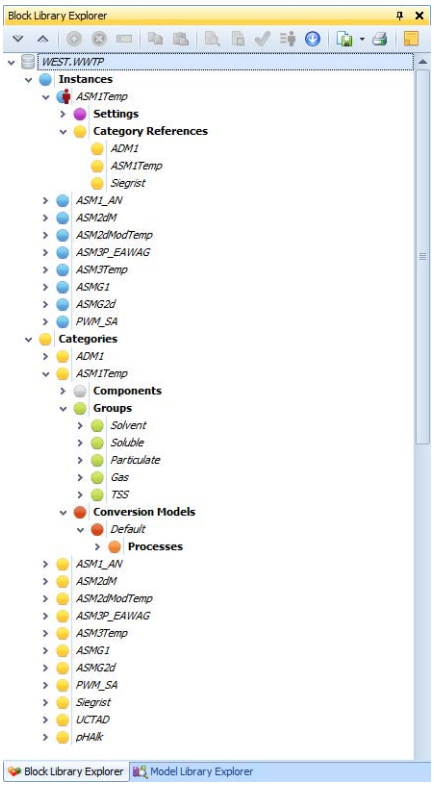


Figure 10.1 Block Library Explorer

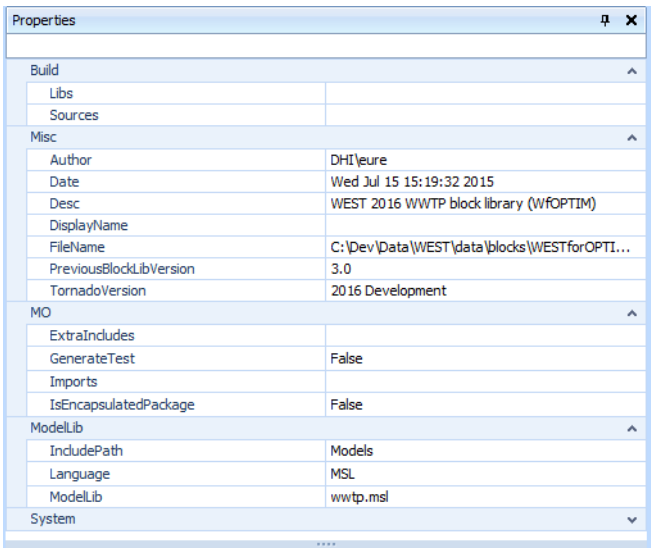


Figure 10.2 Properties pane

10.1 The WEST model Library

It is essential that one clearly understands some very general principles related to the architecture of the WEST model library.

- Model Specification Language (MSL) is the special programming language used for the WEST model library
- A **Model** in WEST is an entity termed CLASS in MSL
  - A CLASS associated to an **icon** will appear in the ClassName drop-down of the Properties pane of that particular Block in the WEST GUI
  - A CLASS can be derived from another CLASS - and thus inherit all the objects defined in that CLASS, i.e. parameters, state and interface variables, as well as its behaviour, i.e. its equations
- The collection of Models of WEST is referred to as the **Model Library**. The WEST Model Library, with the exception of some header files, is generally stored in the Models subfolder of the WEST(forOPTIMIZATION) data folder
- The **Block Library** refers to both the Model- and the Icon Library - the latter stored in the Icons subfolder of the WEST(forOPTIMIZATION) data folder
- The **BlockLib.xml** file is a synthetic representation of the Block Library, which is necessary for WEST during the building phase of a layout.

Key components of the BlockLib.xml are Instances and Categories.

- A **Category** defines the set of **Components** that constitutes the state (or component) vector. The same set of Components may be subject to different sets of Processes - which are termed **Conversion Models**.
  A Category can therefore be associated with a single set of Components but multiple Conversion Models.
- An **Instance** defines primarily the state vector that flows through the Blocks in a WEST layout and the set of Processes that the elements of the state vector (Components) are subjected to, in every Block. As long as suitable *interface models* are provided between Categories, an Instance may use multiple Category (termed **Category References**).
  A Category can therefore be instantiated by more than one Instance.
  Furthermore, an Instance assigns specific values to a set of global variables, termed Settings, e.g. the number of layers in a layered-model such as the Takács model for settling tanks.

10.2 The User Interface

The Block Library Explorer (Figure 10.1) presents a view of the BlockLib.xml file.

The following elements can be identified:

- the name of the BlockLib, "WEST.WWTP" (root of the tree)
- two main branches, Instances and Categories
- the Instances: ASM1Temp, ASM1_AN through to PWM_SA
- the Categories: ADM1, ASM1Temp through to pHAlk
- as an example of multi-category Instance, ASM1Temp has 3 Category References: ADM1, Siegrist and ASM1Temp
- a Category has got three predefined fixed branches:
  - Components: contains the list of Components
  - Groups: standard MSL groups are Solvent (limited to 1 element), Soluble, Particulate, Gas and TSS
  - Conversion Models: can have multiple models; each conversion model defines a set of Processes

Notable properties of a BlockLib.xml are shown in Figure 10.2:

- the description ("Desc")
- the FileName
- the IncludePath: generally the "Models" sub-folder
- the Language: MSL
- the ModelLib, i.e. the rootfile

The file structure of the main folder of the Model Library (the IncludePath in Figure 10.2) is shown in the Files page of the Model Library Explorer (Figure 10.3, Left).

All the standard functionalities of a file explorer are supported: addition/deletion; cut, copy, paste; rename of folders and files.

The file inclusion structure of the library is shown in the Includes page of the Model Library Explorer (Figure 10.3, right).

Notable files of the WEST Model Library are:

- the rootfile: wwtp.msl
- the generic.msl which implements all base data types
- the wwtp.main.msl, the main branch which includes a series of "header" files:
  - general.msl: for base classes of general use in the library
  - definitions.msl: for the definitions of the state vectors (Components) of each base Category
  - reactions.msl: for the definition of the Conversion Process vectors of each base Category

– roughly a header file per unit process (Block), e.g. buffer tanks, activated sludge tanks, primary clarifiers, etc.

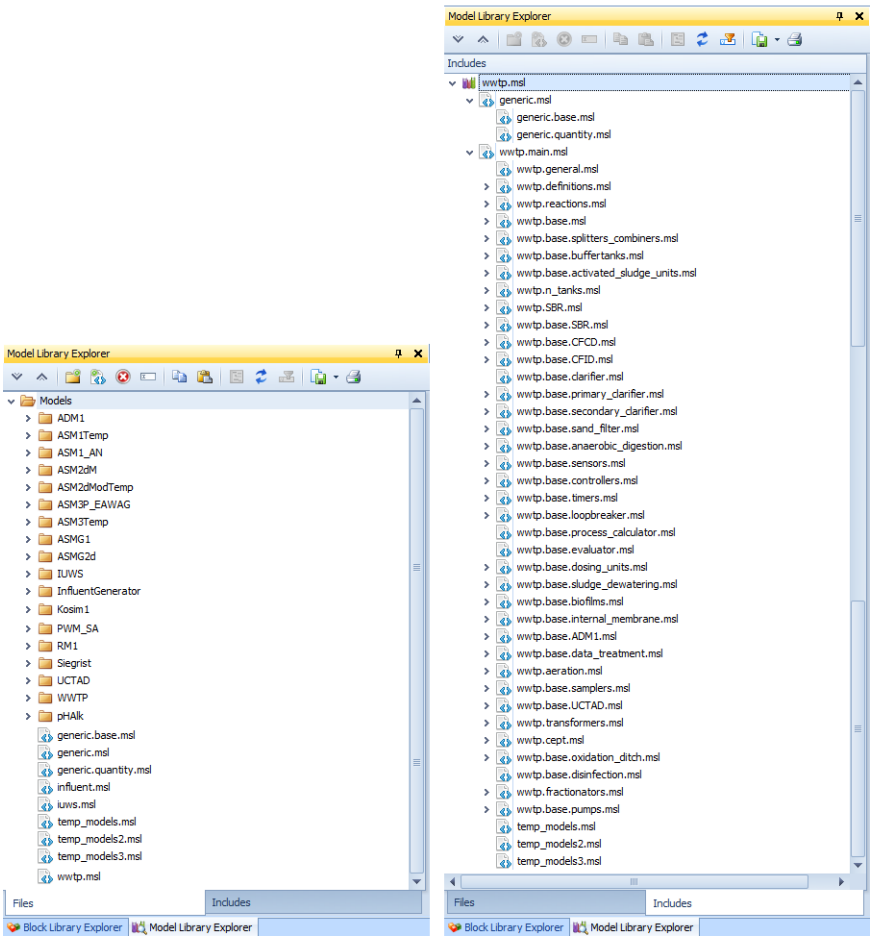


Figure 10.3 Model Library Explorer
Left: file structure, Right: inclusion viewer

10.3 Examples

10.3.1 How to create a new Block Library

- In the Application menu, choose "New" / "Block Library …"
- Provide a suitable name, e.g. "WEST.WWTP"
- Choose the location
- Optionally: provide a description for the library

A new, empty, BlockLib is created, in the Block Library Explorer, with only two branches: Instances (blue circle) and Categories (yellow circle).

Instances and Categories can be populated by creating instances (see below) and importing - if existing, or creating categories (see below).

The key properties of a Block Library are briefly discussed above.

By default, the IncludePath, Language and ModelLib will be set to "Models", "MSL" and "wwtp.msl" respectively.

10.3.2 How to create a new Instance

- In the Block Library Explorer pane: right-click on "Instances" (blue circle, immediately under the root, i.e. the BlockLib's name) and choose "Add …"
- Provide a suitable name
- Default Settings, Category and Groups will be created if the corresponding check-box is selected.
  This may be convenient especially when the Instance/Category specifications (i.e. its Conversion Model) are already available in the model library. See below.

A new Instance is added to the tree equipped with default settings and the reference to a category with the same name. A template for such category is also added under the branch "Categories" - with (empty) "Components", "Groups" and "Conversion Models" sub-items.

10.3.3 How to create a new Category

Three alternative ways are possible:

1. through the creation a new Instance and associated Category template (illustrated above)
2. by copy-pasting an existing Category
3. by creating a new Category from scratch

The only difference between options 1 and 3 is that a Category template, automatically generated upon creation of an Instance, is given the same name as the Instance and the default Groups are added.

Manual creation of a new Category

- Add the relevant Components - one (generally the first) is to be the Solvent
- Define the relevant Groups (only option 3, above)

- Drag each Component to the relevant Group
  - Solvent is to have one and only one Component
  - Soluble, Particulate and TSS are to be populated by one or more Components
  - Components can belong to only one Group
  - Components within a Group are to be adjacent (in the Components list)
- Add one or more Conversion Model
  - Add one or more Processes to each Conversion Model
- Double-click on each Conversion Model node to display (and edit) the corresponding Gujer Matrix

Copy-paste an existing Category

- In the Block Library Explorer, right-click on the name of the existing Category and chose the Copy item from the context menu (or use the corresponding menu button)
- Select the "Categories" branch of the tree-view and chose the Paste item from the context menu (upon right-click) or from the menu
- Make the necessary changes to the new category: you may want to
  - rename the category
  - rename/delete/add new Components and Processes
  - edit the Gujer Matrix

10.3.4 How to import an existing Category

- In the Block Library Explorer pane: right-click on "Conversion Models" (red circle, one level below the selected Category) and choose "Import …"
- Two alternative sources can be selected: "Model Library" or "Block Library":
  - From Model Library:
    - Select the root file: the implementation of the conversion model is to be available in the "Models" subfolder
  - From Block Library:
    - Select the BlockLib.xml file
    - Choose the Category and the Conversion model that are to be imported
- Provide a suitable name: if this is the only Conversion Model for the given category (standard situation for all the base categories, with the exception of "PWM_SA"), it can be "Default"
- Among the "Settings", the "Instance" should be automatically set to the name of the category

- All other fields of the Import dialog (Figure 5) may be left as per default

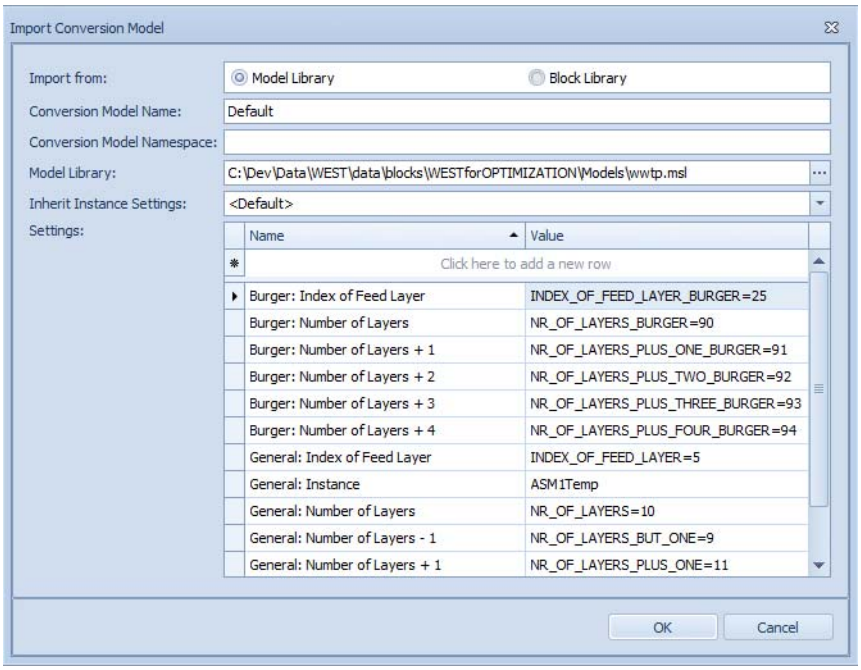


Figure 10.4 Import Conversion Model dialog

This operation can be carried out under one of the following instances:

1. the Category is blank, i.e. prior to adding Components
2. the Category is not blank, i.e. Components are present and Groups are filled in
3. the Conversion Model is to be used as secondary Category Reference for an Instance

Case 1: the Category is blank

By importing the Category specs, all the Components and Groups defined for that Category will be automatically transferred.

Case 2: the Category is not blank

There is some intelligence built in the Import functionality: if Components and Groups exactly match the existing ones, the import will proceed. If not, it will give an error message and abort.

Case 3: secondary Category Reference

Examples in the standard 2020 Block Library are the Siegrist and ADM1 Categories of the ASM1Temp Instance; the ADM1 Category for the ASMG1 Instance; or the UCTAD and pHAlk Categories of the PWM_SA Instance.

Due to some restriction in the MSL language/compiler, there cannot be two enumerations called "Components" within the same instance. The solution is to prefix the secondary Component group with a suitable string and use the same prefix for the relevant indices, such as the IndexOfFirstSolubleComponent.

This is achieved via the "Namespace" property of the Category:

- blank for the main Category Reference
- the name of the Category for a secondary Category Reference: Figure 10.5

Remark: the Namespace property of the Category is to be set prior to importing the Conversion Model.

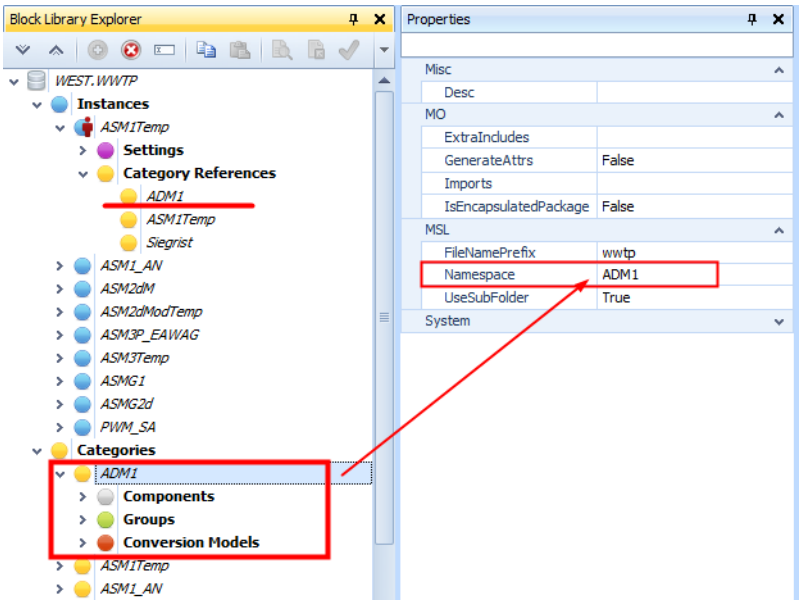


Figure 10.5 Figure 6 Properties for a secondary Category Reference

10.3.5 How to generate code

Code generation, that is the generation of the relevant MSL files, can happen at 4 levels:

- at the level of a Conversion Model
- at the level of a Category
- at the level of an Instance
- at the level of the BlockLib

Code generation for a Conversion Model

- Select the Conversion Model
- Right-click and choose Generate Code - or click on the corresponding button in the Block Library Explorer menu
- Or: in the Gujer Matrix editor, click on the Generate Code button

This will generate the following items:

- a "wwtp.Volume[CATEGORYNAME]ConversionModel.reactions.msl" file containing the Reactions enumeration type
- a "wwtp.Volume[CATEGORYNAME]ConversionModel.body.msl" file containing the body of the Gujer Matrix
- a set of files (.interface, .parameters, .state, .stoichiometry, .kinetics and .sensors) which are included within the .body file

Remark: the .reactions.msl and .body.msl are to be manually included in the library hierarchy.

Code generation for a Category

- Select the Category
- Right-click and choose Generate Code - or click on the corresponding button in the Block Library Explorer menu

In addition to the files generated in the previous case, this will generate:

- a "wwtp.definitions[CATEGORYNAME].msl" file containing the Components enumeration type

Remark: similarly to the previous case, also the .defintions is to be manually included in the library hierarchy.

Code generation for an Instance

It generates code for each Category of that particular Instance.

Code generation for the BlockLib

It generates code for all the Categories instantiated in the Library.

11 Unit Editor

11.1 Description

The Unit Editor is a small application that allows for visualising and managing the WEST unit table. This table is stored in XML format in a file named Tornado.Main.xml that is located in <PersonalFolder>\DHI\WEST\etc after installation of the WEST product. The WEST unit table is used to perform unit conversions within the WEST software, and the UnitEditor allows for adding, removing and modifying entries in this table.

Each unit is part of a category, which refers to the quantity (e.g. Length) that the unit pertains to. Unit conversions are only possible between units that are part of the same category.

For each unit, the following information items can be supplied:

- **Name**:
  Name of the unit (should be unique)
- **Category**:
  Category that the unit belongs to
- **Description**:
  Optional textual description
- **Factor**:
  Factor to be used to convert the unit to the corresponding standard unit in the metric system
- **Offset**:
  Offset to be used to convert the unit to the corresponding standard unit in the metric system
- **Metric Unit Name**:
  Name of the corresponding standard unit in the metric system
- **Imperial Unit Name**:
  Name of the corresponding standard unit in the imperial system

11.2 Quick User Guide

1. After starting the application, three windows appear: Categories, Units and Logging.
2. In the Categories Window, categories can be added (by entering data into the topmost row) and removed (by pressing the Delete key). Removing a category will also remove all associated units.

3. In the Units Window, units can be added (by entering data into the top-most row) and removed (by pressing the Delete key). The Metric Unit Name and Imperial Unit Name items are to be selected from a drop-down box, which implies that only already existing units can be used as Metric / Imperial Unit Name.

INDEX

A

Advanced Virtual Experiments .44, 46, 107
Algebraic Loop 22
Align 49
Align Blocks 166
Analysis Properties 44, 117, 123, 127, 134, 138, ...145
Application Menu 16, 36
Arrange 48

B

Block 20
Block (Fixed Rotation) 160
Block (Flip) 161
Block (Model Selection) 167
Block (Resize) 159
Block (Rotate) 160
Block Editor 13, 48, 215
Block Icon 74
Block Label 75
Block Library 18, 26, 39
Block Properties 72, 158
Block Summary 27
Block(s) 20, 157, 161, 163
BVDF Criteria 145

C

Calculator Variable(s) 104, 171
Causality 216
Check box 25
Classification Criteria 145
Code Editor 13
Combo box 25
Connection(s) 20, 23
Context Menu 50
Control Center 27
Controller 101, 172
Conversion Model 100, 106
Create New Project 153
Criteria 148, 149

D

Data Import 179
Data Path 39
Data Sources 66
De-fractionation Layout 178, 189
De-fractionation Model 54, 106
Derived Variables 138
Description (Project) 44
Design Mode 173
Diagnostics 48
Docking 16
Dynamic 45, 107, 108

E

Effluent Wizard 188
Export 95, 205

F

Field(s) 24, 25, 43, 81, 82, 111, 193
Flip 49
Fractionation Model 53, 99
Frequency 80

G

Gauge 25, 201
Generators 18
Getting Started 17
Getting Started Sheet 15
Gettingg Started Sheet 17
Global Sensitivity Analysis Experiment 107, 127

I

Icon 20, 72, 161, 215
Illustration 42
Influent Generator 53, 55
Influent Wizard 176
Influent/Effluent Wizards 53
Initialise (Simulation) 37, 206
Input and Output Sheets 24
Input Data 54
Input Field 186
Input File 100
Input Model 99
Input Variables 138
Integration 109
Integrator 44

L

Label 20
Language 37
Layer(s) 34, 73, 158
layers 34
Layout Sheet 20
Line 78
Local Sensitivity Analysis Experiment .107,

123
Logging 33
Loop Breaker 22

M
Manipulated Variable 24
Manipulated Variables 100, 138
Matrix Editor 13
MaximumConnections 216
Menu Bar 16
Model Category 99
Model Editor 13, 48
Model Explorer 28
Model Instance 18
Model(s) 167

N
News Headlines 17, 19

O
Objective . 118, 127, 131, 133, 150, 153, 198, ... 207
Objectives 124
Open a Project 155
Orthogonal Connection 51
Output 110
Output (Settings) 44
Output Model 106
Overview 35

P
Palette 215
Parameter 24
Parameter Estimation Experiment 107, 133
Parameters 100, 138
Picture 77
Plant Model 105
Plot 25, 43, 112, 194
Polyline Connection 51
Product 37
Projects 18
Properties 33

R
Radio button 25
Rectangle 76
Replace Icon 51
Rotate 48
Run Criteria 145, 147
Run Dynamic Simulations 206

S
Samples 18
Scenario Analysis Experiment . . 107, 138
Sheets 15, 19
Shortcuts 96
Simulation Output 110
Simulation Properties 44, 108
Slider 43, 80, 185
Slider Pane 79
Sliders 24
Solver 44, 109, 130, 135, 149
Status Bar 16
Steady-State 45, 107, 108
System Windows 16

T
Table 25, 43, 94, 112, 193
Templates 18
Terminals 20, 23, 216
Textbox 76
Time Series Criteria 119, 145
Tools 40
Top-Level Parameters . . . 101, 168, 169
Top-level Parameters 52
Tutorials 18

U
Uncertainty Analysis Experiment . 107, 145
Unit Editor 13, 48, 233

V
Variables 118
Virtual Experiment 27

W
WEST Options 36
WEST Tools 13, 48

Z
Zooming 35, 50