

4. SIMPLE MODEL

This chapter guides the user through the modeling of a simple hydraulic transmission using the BOOST Hydsim Workspace. This exercise does not aim to model a realistic system. Its primary goal is to demonstrate the basic modeling steps with as simple a system as possible.

4.1. Pre-processing Project Structure

For post-processing, result files must be loaded from a specific project structure (lower case).

Create a project folder structure to place models in as follows:

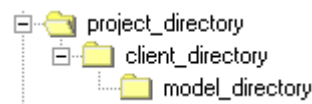


Figure 4-1: Project Structure

Results directories and files are created automatically.

4.2. Creating the Model

First create the system model using icons (elements) and connections (lines). The following steps are necessary to develop models using this BOOST Hydsim simulation software.

The schematic of a simple hydraulic transmission is shown in *Figure 4-2*. It consists of a piston, connected via spring and damper to a mechanical boundary with a defined motion. The motion of the boundary initiates the displacement of the piston in volume 1, filled with fluid. This causes the pressure increase in volume 1. This pressure disturbance propagates with the speed of sound along a line (duct) and within a finite time reaches its output end, which is connected to volume 2. At this instant volume 2 “receives the information” about the pressure change and the fluid starts flowing into it. Frictional losses and unsteady wave effects in the line cause the decaying oscillations of pressure and flow rate in the system.

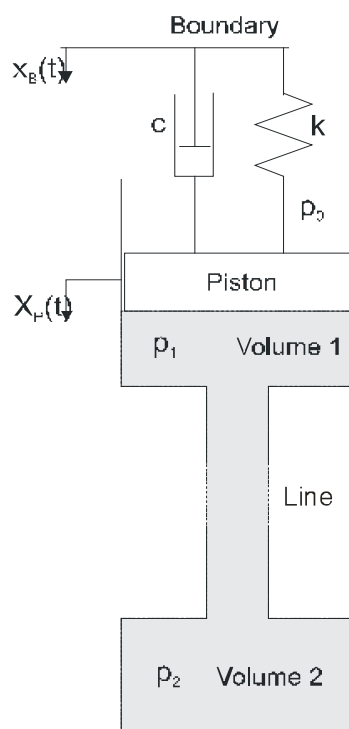


Figure 4-2: Schematic of Simple Hydraulic System

Volume 2 has no outlet, therefore fluid is initially compressed there. This causes the rapid rise of pressure p_2 (due to low compressibility of the fluid) followed by the return flow from volume 2 into line and volume 1. In this way, oscillations of pressure in both volumes, flow rate in the line and vibration of the piston all result, as shown later.

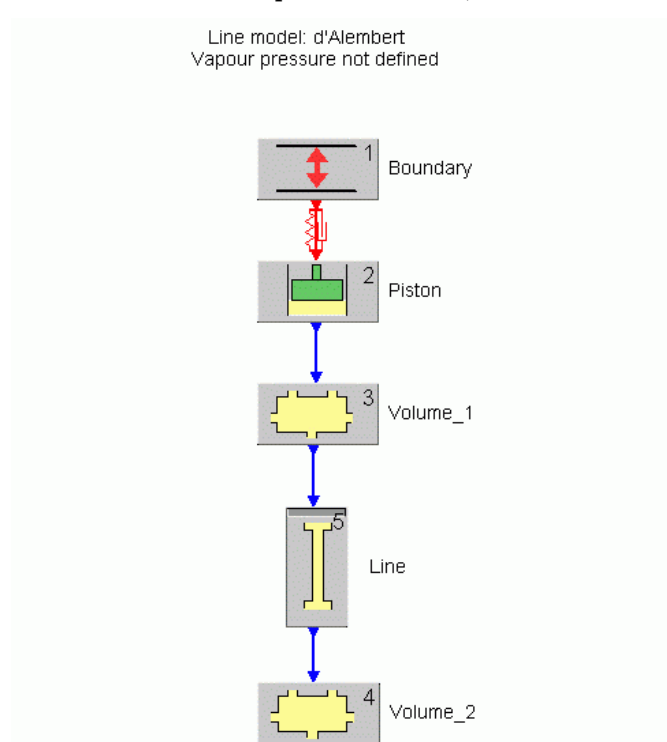



Figure 4-3: BOOST Hydsim Model of a Simple Hydraulic System

The following procedure is used to create the BOOST Hydsim model shown in *Figure 4-3*.

1. Open a new session of BOOST Hydsim.
2. Create the boundary element. Expand **Boundary** in the elements tree by double-clicking or clicking with right mouse button. Then double click **Mechanical** to place a mechanical boundary icon in the workspace window.
3. Create a standard piston from the piston group in the same manner as described. Vertically align this icon below boundary.
4. A mechanical connection between the two elements can now be established. Connect boundary with piston by clicking  in the palette menu. The attachment points appear along the edges of the elements. Note that all points have red color, which means that only mechanical connections are permitted. The number of attachment points indicates the maximum number of connections available for the element. For a mechanical boundary eight mechanical connections can be defined. Additionally, one wire connection can be specified. For a piston, up to six mechanical, two hydraulic, three special and one wire connection can be defined. To connect the icons, place the cursor on the starting point, and drag it to the end point until it forms a circle. Click with the right mouse button to establish the connection.

The completed connection will show as a directional arrow. The connected attachment points can be highlighted with the mouse and dragged along the edge of the icons to a desired position.





5. Continue placing icons and connecting them, in the following order:
 - a) **Volume | Standard** below piston
 - b) **Line | d'Alembert** model below volume. Use keyboard button "r" to rotate the icon 90 degrees clockwise and press spacebar to update the icon title.
 - c) **Volume | Standard** below line
 - d) The rest of the connections are hydraulic and appear as blue lines. To find the connections, click  in the palette menu. The attachment points will appear along the edges of the elements. Connect them in the same manner as before.
 - e) The complete model of the hydraulic transmission on the screen should look like *Figure 4-3*.
6. Once the system model is completed, enter input data by double-clicking the icon with the left mouse button or click the right mouse button and select **Properties** from the pop up menu. Alternatively, click on the icon and select **Element | Properties**.
7. To insert text in the drawing area click  in the palette menu and then click the left mouse button where required. To change font and size, select the text with the left mouse button and then select font and size from the palette menu. Press  for bold text and  for italic.
8. Double-click the **Mechanical Boundary** element to open the following window:

Figure 4-4: Mechanical Boundary Input Dialog

- a) The default name of the element is `Boundary`. This name is user-specified and can have arbitrary length. However, only the first 12 characters will be used in the GIDAS output file with extension `.GID`, and only the first eight characters in the general output file with extension `.dat`. The Impress Chart postprocessor will show the full name in the Result tree.
 - b) Select an appropriate coordinate or velocity in the required direction option for **Variable** - in this case x-coordinate. The title of the second column in the table below will change accordingly.
 - c) The table domain can be Time (default) or Reference angle (refer to Simulation Domain in the Calculation control window - *Figure 4-12*). This is performed by clicking the right mouse button on the appropriate field with unit (below Time/Ref. Angle) and select the desired domain (refer to the unit group **Simulation Period Units**). It should be emphasized that the domain definition applies only to the current element (data table). In addition, a scaling factor for the first column of the table can be activated and specified. After that the user must select which type of boundary condition is to be specified (coordinate or velocity) and in which direction (x, y, or w).
9. To load values in the **Coordinate/Velocity** table, observe the following:
- a) The default for this table is 0/0 in the first line.
 - b) To add values and a new row, click **Insert Row**.

- c) To load values from an existing file, click **Load....** A dialog opens to locate a specific file.
- d) Here, initial and final displacements against time are input to represent the required input motion of the Boundary.
10. **Boundary** (Mechanical, Pressure, Flow Rate, or Hydromechanical) is a particular element of BOOST Hydsim. Contrary to the other elements, it does not represent any equipment but is necessary for connecting the system to the surrounding environment.



Note: You can change the units of input parameters with and without recalculation of their values. To change only the unit of measure, click the left mouse button on the unit field and select the desired unit from the Pop-up menu. The unit will be replaced but the actual value will stay unchanged. To change the unit of measure and recalculate the parameter value, click right mouse button and select the desired unit. Now the new unit will appear and parameter value will be converted from the previous unit into actual one (refer to *Section 3.5* for default input units).

11. In the next steps, the input data for all the selected elements is entered, beginning with the **Mechanical Connection**. Double-click the icon to open the window shown in *Figure 4-5*.

Figure 4-5: Input Dialog of Mechanical Connection

- a) The default name is Spring/Damper. Default direction is x. In x and y directions (translational) the preload force, linear stiffness and damping can be specified. In w direction (rotational), if selected, the preload torque, torsional stiffness and

damping must be defined. Click on two different axes in the dialog box and the names of the required input parameters change accordingly.

- b) If the connecting Spring/Damper is nonlinear, its characteristic has to be defined in the table after selecting the **Variable** toggle button. For variable connections, preload and stiffness as a function of relative displacement (angle) can be specified. Damping can be defined as a function of displacement or velocity.

12. The **Standard Piston** is one of the most widely used elements in hydraulic systems. The input dialog of **Standard Piston** element is as follows:

The screenshot shows the 'Piston 2 - Hydraulic Piston' dialog box. It includes fields for Element Name, Rigid/Elastic Body selection, Moving mass, Coulomb friction force, Piston stroke, Cross-section (Area/Diameter), Piston diameter table, Stop cross-section, Stop area table, Stop model selection, Piston stop data table, and Direction of Element motion table.

Figure 4-6: Input Dialog of Standard Piston

- a) The input parameters of the **Piston** element are:
- Moving mass
 - Coulomb friction force
 - Piston stroke
 - Cross-sectional area/diameter at input and output end, respectively
 - Cross-sectional area/diameter of input and output stop, respectively (refer to [BOOST Hydsim Users Guide](#) for definitions)
 - Stiffness and damping parameters of piston stops at input and output end.

For mechanical springs, moving mass is defined as the piston mass plus 33% of the total mass of the attached springs (connections). If the connecting stiffness is high and the mass is low, high natural frequencies will occur in the system, which may

cause numerical problems for the integration. To avoid numerical instability in this case, a very small **time step** should be specified in **Simulation | Control**.

- b) The stiffness and damping parameters of the piston stops are used in the calculation only when the piston reaches the input or output stop. These parameters must have finite values to prevent numerical discontinuities. Usually, the stiffness of piston stops should be several orders higher than the corresponding stiffness of the mechanical connection.
 - c) The piston element may have up to six mechanical connections (with other elements). Furthermore, two hydraulic connections can be specified: one on the input end and one on the output end. All connections can be defined only in the x -direction.
13. The next element, **Standard Volume** is one of the most common hydraulic elements. The input dialog is as follows:

Figure 4-7: Standard Volume Input Dialog

- a) The default name of the element is `Volume`. As there are two volume elements in this system, it is useful (but not critical) to change the name of the current volume element to, for example, `Volume1`. BOOST Hydsim recognizes elements by their ID numbers (placed at the right topside of the icons), therefore elements with the same name do not cause any problem for calculation. However, this may lead to confusion in post-processing because the user may not be able to distinguish easily between the output of two elements with the same name.



Note: To change the icon name, click on it with the left mouse button and then click **A**. Click **B** for bold and *I* for italic. The two input parameters of the Volume element are the volume at start of calculation and vapor pressure. If the volume element is linked to other mechanical elements, which exhibit displacements (piston in this case), then the volume itself is obviously a variable changing with time.

b) Pressure within a volume is considered to be same at each point. In BOOST Hydsim, pressure is treated similarly to the mechanical stress and formally may be also negative. To prevent negative pressure, vapor pressure (positive value) must be specified. By default it is set to 0.001 bar (100 Pa). If vapor pressure is defined, cavity effects will be considered in the **Volume** element. This implies that if the pressure drops down to the vapor pressure, it will be kept constant at this pressure value and (to preserve a mass conservation law) a vapor cavity will be calculated. Before the pressure in the volume may increase again, this cavity must be refilled with fluid. If the vapor pressure is set to 0 or negative value, the cavity effects are neglected and negative pressures are possible. Results for both cases will be discussed later in this section.

c) Select **Fluid Properties** by opening the following window.

Volume_1 3 - Fluid Properties

☐ Local fluid properties from Property Database
 Fluid name:
 Initial temperature: K

☒ Global (constant or variable) fluid properties

☐ Local (constant) fluid properties
 Bulk modulus: N/m²
 Reference density: kg/m³
 Kinematic viscosity: mm²/s
 Surface tension: N/m
 Reference pressure: Pa

Description:

☐ Local (variable) fluid properties

	Pressure Pa	Bulk modulus N/m²	Density kg/m³	Kin. viscosity mm²/s	Surface tension N/m
1	0	0	0	0	0
2					
3					
4					
5					
6					
7					
8					

Buttons: Insert row, Remove row, View..., Load..., Store...

☐ Calculate bulk modulus internally from density-pressure function: $E = \rho \left(\frac{\partial p}{\partial \rho} \right)_T$

OK, Cancel, Help, Accept

Figure 4-8: Input Dialog of Local Fluid Properties

14. This input dialog box has the same form for every hydraulic element. The four options for fluid properties are: global, local from Property database, local (constant) and local (variable). Global fluid properties (constant or variable) are defined for the entire system from the **Model | Fluid Properties** menu and will be discussed later in this example. By default, fluid properties are always set to global. To specify element fluid properties differently from those defined in the global table, click the **Local** fluid properties from Property Database, **Local (constant)** or **Local (variable)** fluid properties. Depending on the selection, the respective fluid properties (bulk modulus, density, kinematic viscosity and surface tension) source will be activated. If not chosen from Property database, these values have to be entered. Note that local fluid properties apply only for the actual element.




Note: In BOOST Hydsim, a **Volume** element plays a particular role. To build up a consistent model of your system, hydraulic elements (e.g. lines) are internally connected through volumes because they need pressure as a boundary condition on both sides (refer to [BOOST Hydsim Users Guide](#) for more information).

Volume elements may have up to 10 hydraulic connections and one wire connection. Hydraulic connections are automatically assigned to x direction by GUI. Using other types of connections with a volume element is not permitted by the GUI: if you try to establish it, an error message “Incompatible Connection” will pop up.

15. The Line element is used to represent any kind of pipe, duct, tube or hose. In BOOST Hydsim, clicking on the line icon in the element menu opens a tree with five options: d’Alembert Model, Laplace Transform, Characteristics Method, Godunov Model and MacCormack/Two-phase. These may be used to represent the same physical equipment (pipe). However, the models of each line type are different, getting more and more complex from d’Alembert line to MacCormack/Two-phase. Basically, the d’Alembert model is an analytical solution of the line equation without friction. It uses an empirical damping function and cannot account for non-stationary increase of friction. Laplace Transform is a semi-analytical solution of the line equation which considers non-stationary frictional losses. Method of Characteristics and Godunov method are numerical schemes for the solution of the wave equation. Godunov model is the more advanced single-phase line model because it can work with variable velocity of sound. However, it may require long calculation times. MacCormack/Two-phase line model has higher order accuracy and can be used to model two-phase flow (cavitation). Godunov and MacCormack line models can use different models for frictional losses. The line models are discussed in detail in the [BOOST Hydsim Users Guide](#).

In this example, the simplest model is used: a line based on the d’Alembert solution (with empirical damping function). On the left side of the **Line** icon, there is a small, empty, i.e. not filled with any color, vertical bar, which implies the simplest model option and also applies for other elements (e.g. orifices, nozzles). In general, the amount of red color within the bar provides a complexity measure of the element model. All line icons have the same appearance and name as the **d’Alembert Line** icon. However, the bar in the **Laplace Transform** icon is one-third filled and the bar in the **Characteristics Line** icon is two-thirds filled with red color. This does not mean that the line solution by the method of characteristic is the most complex one. In fact, a semi-analytical **Laplace Transform** is much faster but the results are available only at line ends. The **Characteristics** module solves the wave equation numerically by the well-known method of characteristics and thus can provide pressure output at any cross-section along the line and not only at line ends. The most complex single-phase line model is the **Godunov** method. It solves the wave equation by the 1D finite volume method and can work with strongly variable fluid properties. The **MacCormack** line model uses the second-order finite difference method. Combined with bubble dynamics theory, it is applicable for the solution of two-phase flow (cavitation).



Note: Delete unwanted elements or connections by first highlighting the element/connection. Select **Edit | Delete** or click  button in the Palette menu. The element/connection will be erased. Please note that **Undo** option is not available.

16. The input dialog box of the **d'Alembert Line** is as follows:

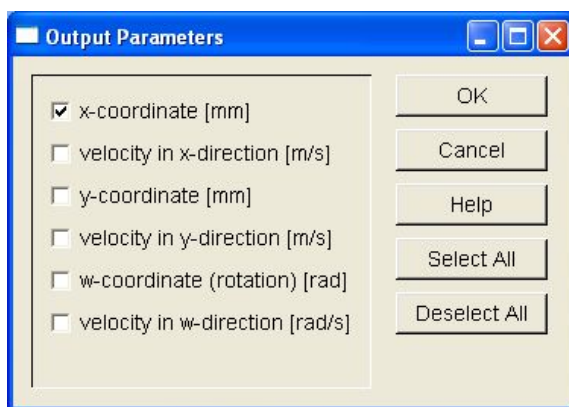
Figure 4-9: Input Dialog of d'Alembert Line

- a) The default name of the element is **Line**. The required input parameters are line length, hydraulic diameter and the exponent of pressure-pulse damping. The latter must be negative as it defines the rate of exponential pressure decay along the line (for more detailed information, refer to the [BOOST Hydsim Users Guide](#)). The Fluid Properties button is also available as for any hydraulic element.
17. The last element of the hydraulic system is **Volume**. It is the second volume element in the system, therefore we call it **Volume_2**. The input dialog of **Volume** element is shown in *Figure 4-7*. **Volume_2** has only one connection: inlet line, thus due to the forward piston motion the fluid will be compressed. If not specified otherwise, the compressibility of the fluid in BOOST Hydsim is treated according to linear acoustic theory.

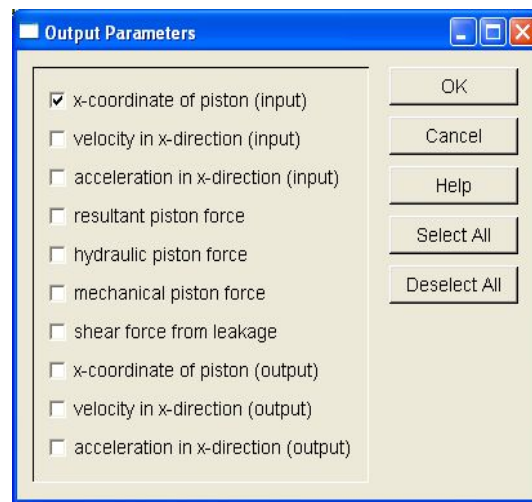
4.3. Output Parameters

After all input data has been entered, the output parameters for each element must be selected. The output dialog associated with each element is opened as follows:

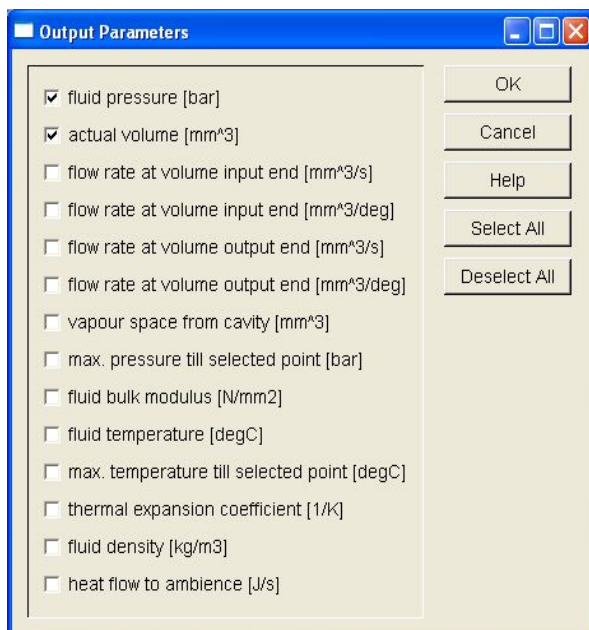
1. Select the element with the mouse.
2. Select **Element | Store Results** to open a window (as shown in *Figure 4-10*) where the appropriate button must be selected so the corresponding variable is stored as output. Selecting **Select All** will store on file all available output parameters listed in the dialog. These can be viewed with the Impress Chart post-processor.



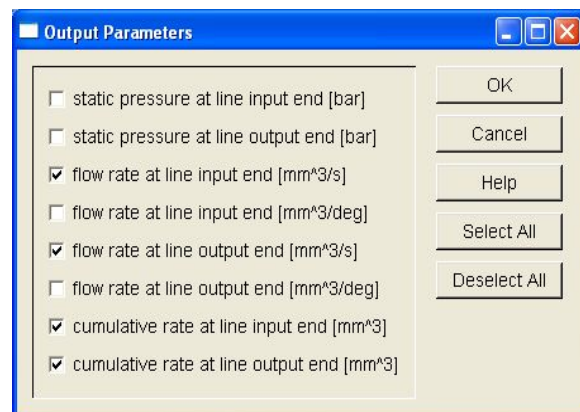
Mechanical Boundary Output Data



Standard Piston Output Data



Standard Volume Output Data



d'Alembert Line Output Data

Figure 4-10: Output Data Dialogs

For the **Mechanical Boundary** as well as **Pressure, Flow Rate and Hydromechanical Boundary**, the output data is the same as the input data because boundary conditions are not influenced by the dynamics of the system. The displacement (x-coordinate) is selected to be stored on the output.

Store the motion (x-coordinate) of the **Piston** on the output. For volume elements, pressures are usually of primary importance, therefore, store them on output. In the output dialog of **Line** element, store the flow rates (volumetric and cumulative).

3. In BOOST Hydsim, a set of initial conditions can be specified for every element except specific groups (Boundary, Cam, Throttle, Orifice, etc.). The Table of Initial Conditions for an element is opened by highlighting this element and then selecting **Element | Initial Conditions**. The Table of Initial Values for piston-type elements has the general form:


Initial Values		
	Coordinate	Velocity
	m	m/s
input x		
output x		

Figure 4-11: Table of Initial Values for Piston, Plunger or Needle

Enter two variables (initial coordinate and velocity).

For volume elements, the Table of Initial Values has only one initial value: pressure. Set initial pressure for Volume 1 and Volume 2 elements to 1 bar. All non-specified initial values are automatically set to zero.

For line elements, the Table of Initial Values requires two variables (initial Flow Rate on input and output side), both in x-direction (flow direction).

4. Next, define the control data for running the calculation by clicking **Simulation | Control** or press .

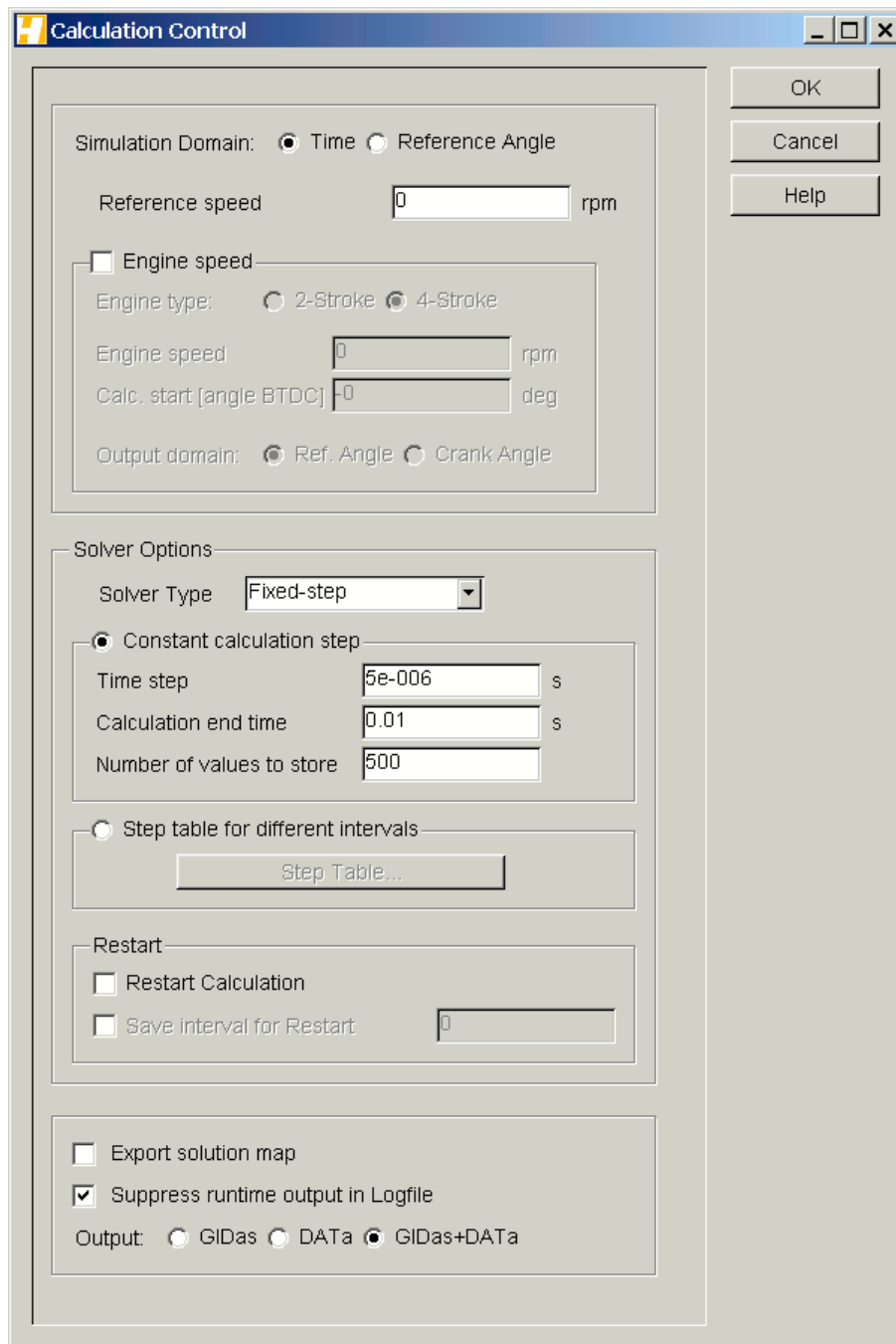


Figure 4-12: Calculation Control Dialog

- a. Select **Time** or **Reference angle** for **Simulation domain** . Our system is not referenced to any rotational speed, therefore we leave the default selection Time. Note that if Reference angle is chosen, a Reference speed must be specified, otherwise BOOST Hydsim will issue an error while starting calculation (after pressing Run) or whilst performing other operations.

If operating in the Reference angle domain, default Reference Angle Step of (cam speed x 10^{-6} degrees cam) can be used. Typically, the Time Step (recalculated from Reference Angle Step using Reference Speed) should not lie outside the limits $10^{-8} < \text{step} < 10^{-6}$ seconds, because instability or an excessive calculation time are likely to occur.

- b. Set the Solver Type to **Fixed-step**. This is a constant time step explicit solver based on a Runge-Kutta-Gill 4th order scheme.

Enter the **Time step** for numerical integration (constant in this case), **Time interval** and **Number of values to be stored**.


Note also that, if a 'coarser' calculation is required before and/or after a key area in the total calculation e.g. the angle over which injection occurs, different Time/Reference Angle Steps, Time/Reference Angle Intervals and Numbers of Values for output can be selected under **Simulation | Control | Step table for different intervals**. This gives the user the opportunity to speed up the calculation in areas of less importance but should not be used unless calculation speed is a problem.

- c. Optionally, **Save interval for Restart** can be entered. If it is specified (value between 0 and Time interval), a binary start-file `simple_line1.STA` will be created on each Saving interval step. It contains the data necessary for restarting the calculation from the selected position.
Restart Calculation allows an already completed calculation to be continued. The new calculation starts with initial conditions taken from a restart file.
 - d. Alternative solver type is **Variable-step** solver. It is a multi-step implicit solver based on Backward Differentiation Formula method (detailed description is given in the Users Guide). In Section 5.2 an example with **Variable-step** solver is presented.
 - e. Within the **Calculation Control** dialog box, activate the **Export solution map** option to create an additional text file `simple_line1.SMP`. This file contains the integration history and might be useful for checking the convergence and tracing back numerical errors. However, understanding of it requires certain knowledge of the integration algorithm and is of no use for an inexperienced user. Moreover, for larger systems, it will occupy a considerable disk space.
4. Select **Model | Fluid Properties** to open the following window:

Figure 4-13: Input Dialog of Global Fluid Properties

Global fluid properties can be specified directly in the dialog or selected from the Property database. Fluid properties can be either variable or constant. **Constant Fluid Properties** (bulk modulus, density, kinematic viscosity and surface tension) are default. Click **Variable Fluid Properties** to activate the Variable Fluid Properties. If variable fluid properties are selected, BOOST Hydsim will automatically assign variable properties from the table to every hydraulic element of the system. In the Table of Variable Fluid Properties, enter the bulk modulus, density, viscosity and (optionally) surface tension as a function of pressure. Remember that for each hydraulic element global fluid properties can be substituted by the local. This might be useful, for example, in taking into account temperature variation along the hydraulic system or in systems containing several fluids.

4.4. Running the Calculation

1. After all input, output and calculation control data has been entered, calculations can be started by selecting **Simulation | Run...** or press . If no error message appears on your screen, the entered data is formally acceptable and the calculation will be started. If GUI detects any problem during straightforward compatibility checks, it will immediately report it on the screen.
2. Under **Task Information** window the basic run-time information is shown (including warnings and error messages) (*Figure 4-14*).

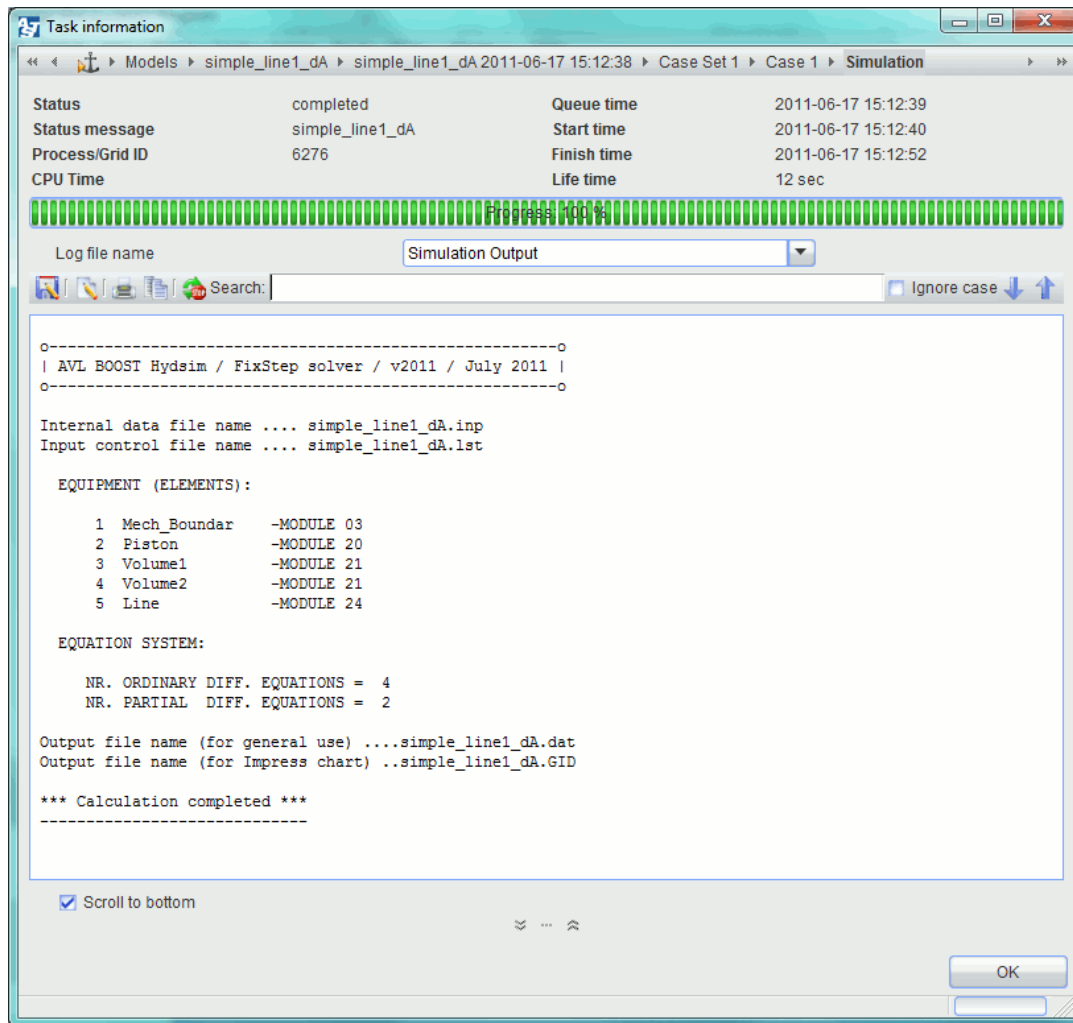


Figure 4-14: Task Information Window

Information about input and calculation errors, the created files, system units and calculation process is available. If no error message is produced and the following

Status: completed

appears in the **Task Information** window, the calculation is successfully completed and all results are automatic stored on the output defined in **Calculation Control**.

- Data output file name (general) simple_line1_dA.dat
- GIDAs output file name (for IMPRESS Chart) simple_line1_dA.GID

3. To compare different line models, it is useful at this stage to substitute a **d'Alembert line** by the next line element **Laplace Transform**. For this, the user has simply to delete the **d'Alembert line** element and its connections to Volume1 and Volume2 elements (using **Cut** option from **Edit** menu) and insert a **Laplace Transform Line** in its place. The new hydraulic connections to volume elements have to be reestablished. The input dialog of new line element is as follows:

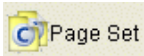
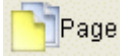

Figure 4-15: Input Dialog of Line (Laplace Transform)

- The default element name is **Line**. The input data in this case is only **line length** and **hydraulic diameter**. Save this change under a different file name, for instance `simple_line1_lp.hyd` and restart the calculation.

4.5. Calculation Results

To view the calculation results, select **Programs | IMPRESS Chart** or select **Simulation | Show Results** to open Impress Chart (post-processor) as shown in Figure 4-16. **Show Results** directly opens the actual model results directory (refer to *Section 6.3.2*).


4.5.1. Create Report

- Select the **Report** tab, click on  and then . Multiple pages and page sets can be created in this manner. Highlight the page to insert the results on.
- Select the **Results** tab, click  or click on the **Results** folder with the right mouse button and select **Load** from the submenu. From the Project directory dialog select the `AllResults` file under the directory `.../Examples/BOOST Hydsim/...` and all BOOST Hydsim result files in the default directory will appear.

Note: Selecting appropriate directory from Project directory dialog box (e.g. `simple_line1.Case1`) and then select file `results.ppd`, will load only the results of the desired example.



Note: Notice that results are loaded as `results.ppd` and as part of the actual model tree from the directory where the models are stored. In many cases it might be inconvenient, so it is recommended to store model files in a specific subdirectory `.../BOOST Hydsim /<model name>.hyd`.

3. Select the type of coordinate system for displaying the results ( Layer for Cartesian graph). Refer to the IMPRESS Chart / PP3 section of the [GUI Users Guide](#) for directions on how to change the titles of the axis, resize the graphs, etc.

Highlight the graph in which you wish to place the results.

Go to the **Results** folder and double click on the results that should be placed in the graph. In this manner, you can place more than one result in one graph.

The custom designed window with main calculation results is shown in *Figure 4-16*.

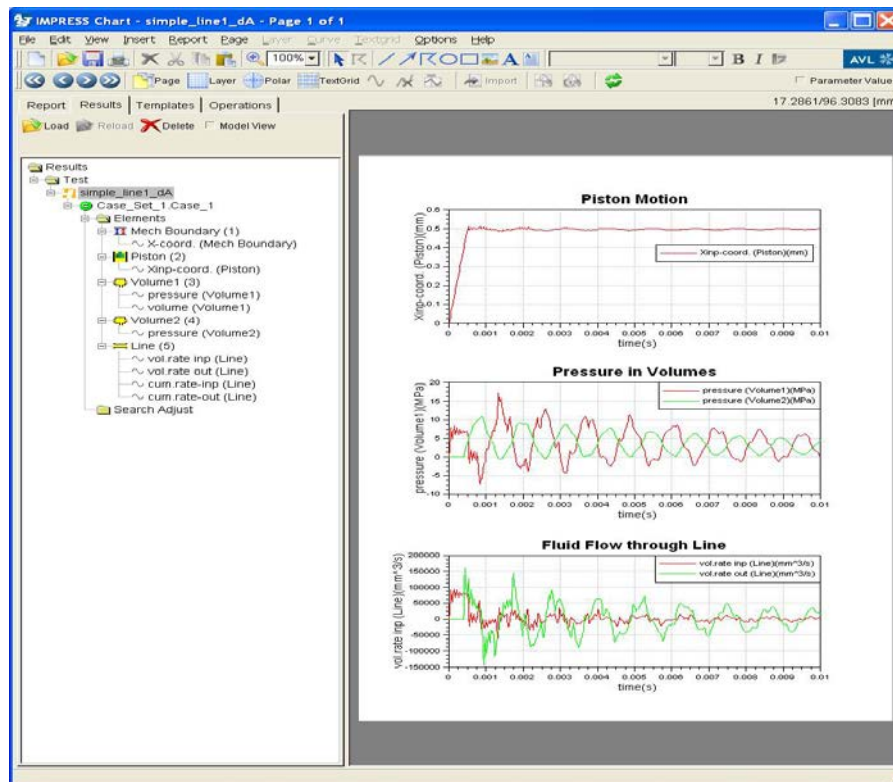


Figure 4-16: Calculation Results without Cavitation Model (zero vapour pressure)



Note: When more than one curve is placed onto the graph, the y-axis title remains unchanged (equal to the title of the first curve placed in the graph).

There is one important difference between the Preview (preview a curve without creating a layer) and user-defined layer (*Figure 4-16*). In a created layer, the new curve will be added to the existing one(s) as long as the legend title of the new curve (parameter name plus element name) is different from the legend titles of existing graphs. Otherwise (if the legends coincide) the respective curve will be overwritten.




Note: Files simple_line1_dA.ppd and simple_line1_lp.ppd are only control files. They search for the files simple_line1_dA.GID and simple_line1_lp.GID from which the actual results are taken. Thus, GIDAs output has to be defined in the **Calculation Control** dialog.

4.5.2. Load GIDas Files

Another method to load results is to load files in GIDas format (*.GID files). Click on the **Results** folder with the right mouse button, and select **Load GIDAS File** from the Pull Down menu. For our example `simple_line1_dA.GID` from `simple_line1_dA.Case_Set1.Case1` directory has to be loaded. In this case, each folder in the output tree is an individual curve. The folder name contains the user-defined element name (up to 12 characters) and program-defined parameter name (with a symbol of a curve in front). By clicking on any folder, the curve will be displayed in the same manner as in the case of **Create Report** (refer to *Section 4.5.1*).

4.5.3. Import Feature

The main purpose of Import feature is to plot specific diagrams (for example volume vs. pressure etc.). To do this, highlight the graph in which you wish to place the results and click on the  **Import** button from the Palette menu. Select the appropriate GIDas file from the Project directory dialog box (e.g. `simple_line1_dA.GID`) and the following dialog box will appear:

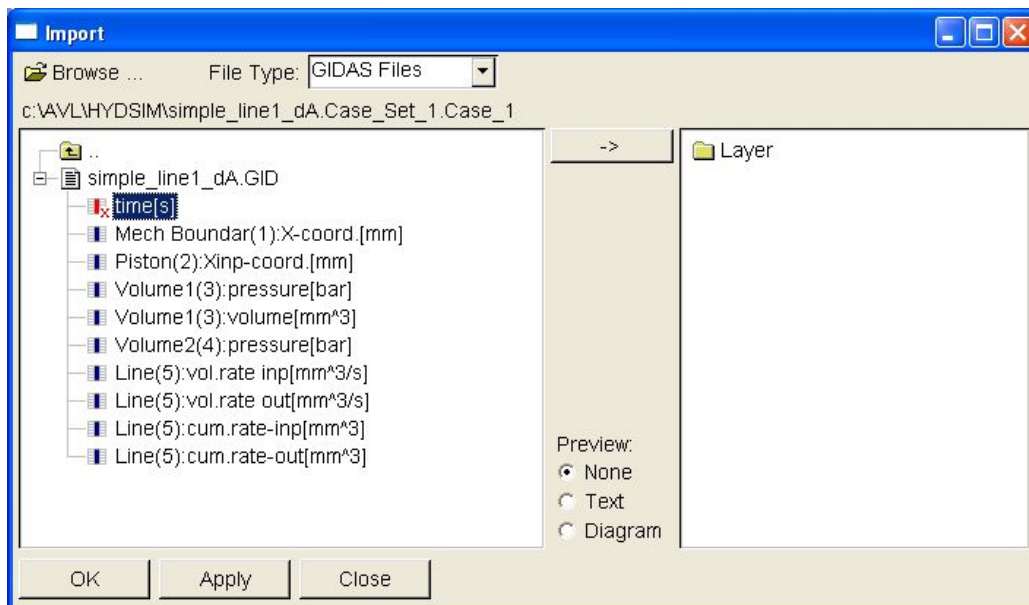



Figure 4-17: Import Window

Set desired item for X-axis by clicking the right mouse button. In our example, select `Piston(2):Xinp-coord.[mm]` item and from pop-up menu select **Use as X** parameter.

Either select `Volume1(3):volume[mm^3]` parameter and press  to import Y-axis or double click on desired parameter to import it as Y-axis. Press **Apply** button to confirm. Create the 2nd layer and select there `Volume1(3):pressure[bar]` for X-axis and `Line(5):vol.rate inp[mm^3/s]` for Y-axis. The following graphs will be created:

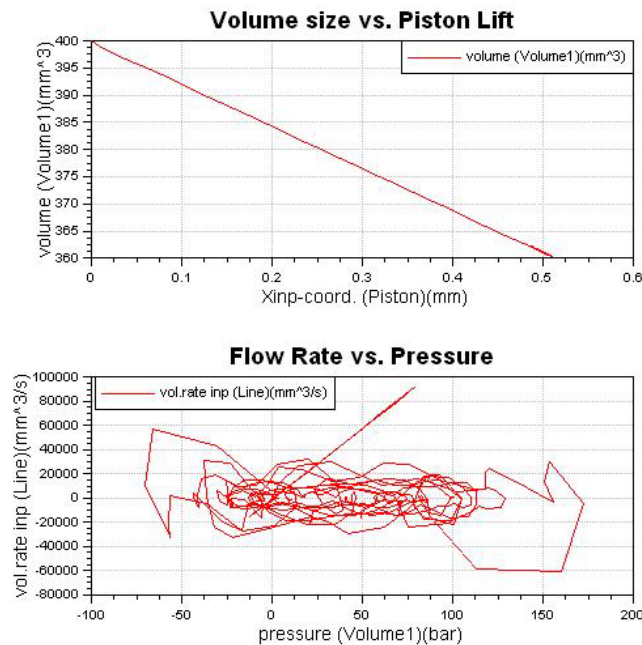


Figure 4-18: Specific Diagrams with Different Variables for X-axis

Note that selection of X-axis variable has to be done with care, otherwise the resulting curve may look very unusual as shown in the 2nd graph of *Figure 4-18*. After results are loaded (.ppd or .GID file) and graphs plotted, select **File | Save as** and save as .../.../<name>.pp2 file (e.g. .../BOOST Hydsim/simple_line1_dA.pp2).

4.6. Results with Cavitation Model

Figure 4-16 shows the calculation results of simple hydraulic system without the cavitation model. In this way, at certain time instants the pressure in volumes gets negative (due to wave propagation in the line). In a real physical system this cannot happen because at certain (low) pressure the fluid starts evaporating (cavitating). To consider this effect, a simple cavitation model is used. For activating it, the vapour pressure in volume elements has to be specified (set to a positive value, default 100 Pa or 0.01 bar) as shown in *Figure 4-19*. If the pressure in a volume drop down to the vapour pressure, it is kept constant and vapor cavities are calculated (they can also be provided on output). Of course, this affects the dynamics of the system. The numerical integration in this case may become less stable, therefore care has to be taken in specifying the appropriate time step in the **Calculation Control** dialog.

Figure 4-19: Input Dialog of Standard Volume with Vapour Pressure



Figure 4-20: Calculation Results with Cavitation Model

Calculation results with cavitation model are shown in *Figure 4-20*. Obviously the pressures and flow rates there look somewhat different from the results of the model without cavitation. Pressures in volumes do not drop below the specified threshold value (0.001 MPa), i.e. they always stay positive. At vapour pressure the cavity (vapour space) is calculated (refer to the [BOOST Hydsim Users Guide](#) for more information).

In the same manner, the user can perform calculation and plot the results of the simple hydraulic system with other line models. Next line model (most common in use) is the so-called Laplace Line available in the examples `simple_line1_lp.hyd` and (without cavitation model) and `simple_line2_lp.hyd` (with cavitation model). We leave this task for the user. Note that results with Laplace and other (more complex) line models can somewhat differ from those for the model with d'Alembert line shown in *Figure 4-16* and in *Figure 4-20*. This is natural, because the Laplace Line contains another (optional) model for frictional losses than the d'Alembert solution with an empirical loss function. However, the user should be aware that this is only an introductory test example, not aimed to gain physically meaningful results.

4.7. Assigning Parameters

Open previously discussed example `simple_line1_dA.hyd`.

Assigning new Parameters is enabled either by means of element input dialog box or global model dialog.

First we will cover assigning through the input dialog box.

To assign Parameters, open the input dialog data box of Volume_1, move the mouse arrow to the description of the initial volume (*Figure 4-21*) and then press the right mouse button.

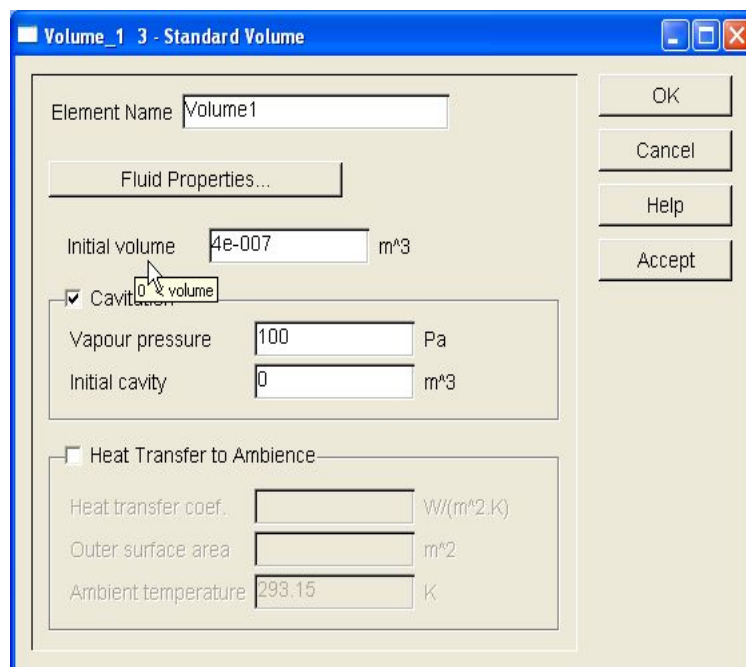


Figure 4-21: Assigning Parameters

The following menu will pop up:

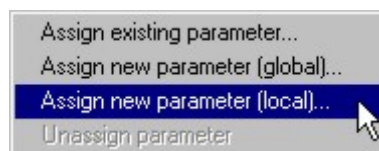


Figure 4-22: Assign Parameter Menu

After selecting either **Assign new parameter (global)** or **Assign new parameter (local)** you will be prompted for the name of the new parameter.

Select **Assign new parameter (global)** and name it e.g. “VOLUME”. This name will replace the original input value in the dialog box, and the parameter VOLUME will get the value of the original input (400 mm³ in this case).

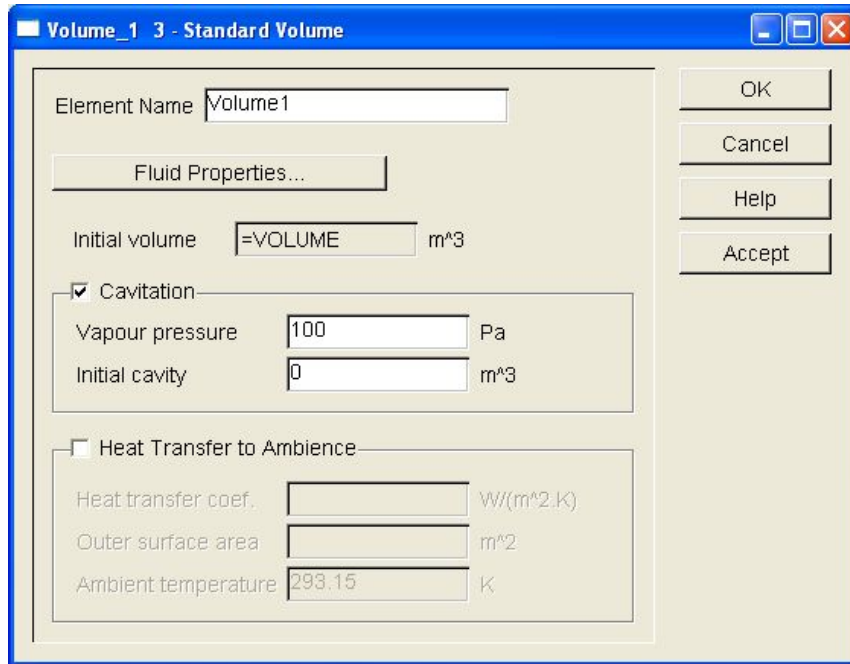


Figure 4-23: Dialog with Assigned Parameter (Variable)

In the same manner, assign local parameters for the Line element and name them as follows:

- Line length: “Length”
- Hydraulic diameter: “Diameter”

The next element for assigning parameters is Piston. Open its Input data dialog box and assign parameters for Piston diameter (mm).

Notice that the Piston diameter has two entries: **input end** and **output end**. In this case assigning parameters has to be performed in the following manner. Click the mouse pointer into the desired input field (e.g. output end) and the arrow will appear in its right bottom corner. Press left or right mouse button on the arrow for the Assign Parameter Menu (*Figure 4-24*), assign it as a local parameter and name it “OUTPUT_DIAM”. In the same way a parameter could be assigned for the diameter of **input end**. Confirm changes with the **OK** button.

Piston 2 - Hydraulic Piston

Element Name:

☒ Rigid Body ☐ Elastic Body

Moving mass: kg

☒ Coulomb friction force: N

☐ Piston stroke (max.lift): m

Cross-section (under pressure)

☐ Area ☒ Diameter

Piston diameter

		input end	output end
Diameter	mm	<input type="text" value="0"/>	<input type="text" value="10"/>

☐ Stop cross-section (under pressure)

☒ Stop Area ☐ Stop Diameter

Stop area

		input stop	output stop
Area	m ²	<input type="text" value="0"/>	<input type="text" value="0"/>

Piston stop data

		input stop	output stop
Stiffness	N/m	<input type="text" value="2e+008"/>	<input type="text" value="2e+008"/>
Damping	N.s/m	<input type="text" value="200"/>	<input type="text" value="200"/>

Context Menu:

- Assign existing parameter...
- Assign new parameter (global)...
- Assign new parameter (local)...
- Unassign parameter...

Buttons: OK, Cancel, Help, Accept

Figure 4-24: Assigning Parameters for a Vector/Matrix Element

After parameter assignment it is possible to view Input Dialog boxes either with parameter names or parameter values. To choose the desired option, click in the check box

☒ **Parameter Values** in the right upper corner of the BOOST Hydsim window. Another way is to click the right mouse button on the right end of the Input Dialog window and check **Show Parameter Values** in the Pop-up menu (*Figure 4-25*).

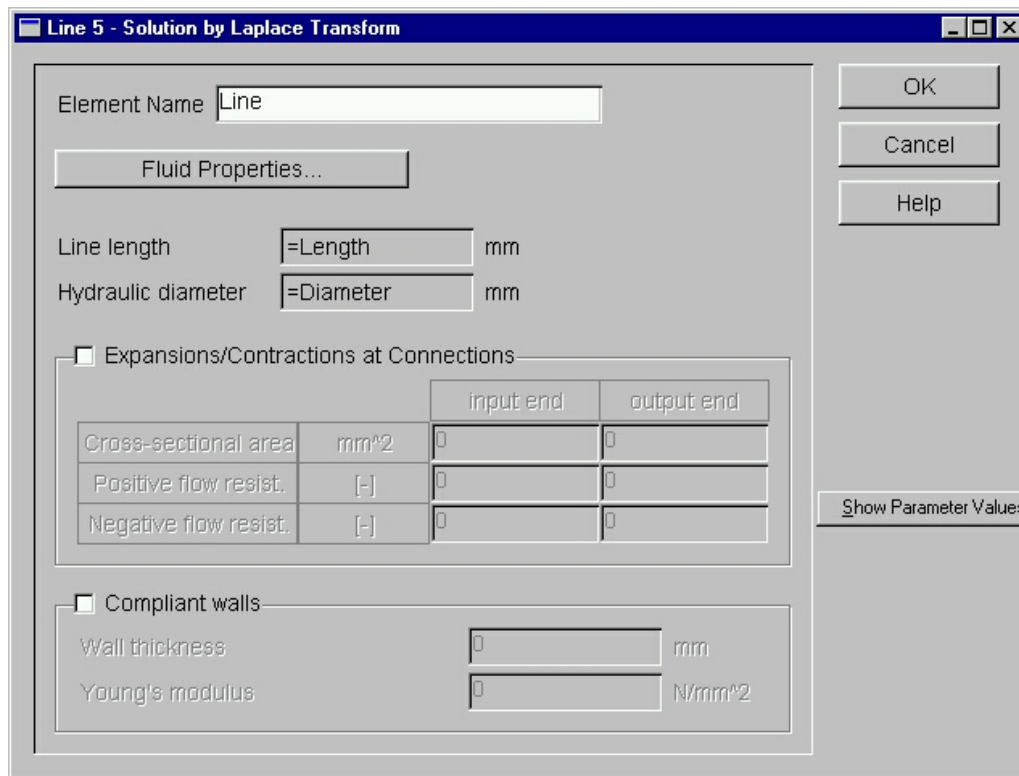


Figure 4-25: Show Values of Assigned Parameters

4.7.1. Assigning Existing Parameters

To assign an existing parameter, open Assign Parameter Menu (*Figure 4-22*) and select **Assign existing parameter**. The following dialog box will pop up:

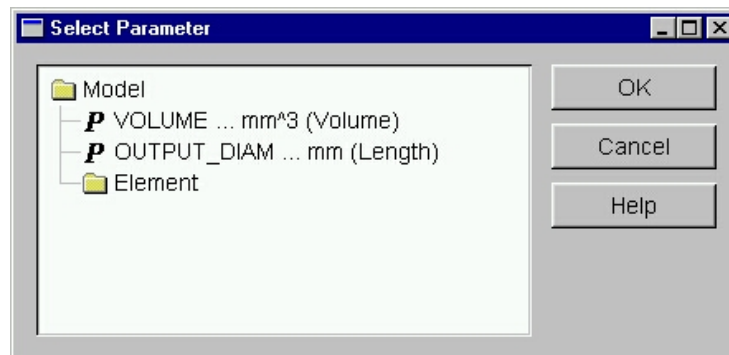


Figure 4-26: Select Parameter Dialog

All global parameters are available for selection but only the parameter with the appropriate unit of measure will be accepted. For instance, the only acceptable parameter for initial volume in Volume2 dialog is “VOLUME”.

4.7.2. Unassigning Parameters

To unassign an existing parameter, open the Assign Parameter Menu (*Figure 4-22*) and select **Unassign parameter**. The actual parameter value will appear in the input field.

4.7.3. Model & Element Parameters

After assigning parameters, you can edit the actual values of all assigned parameters by selecting **Parameters** from the **Model** menu. The following dialog will appear:

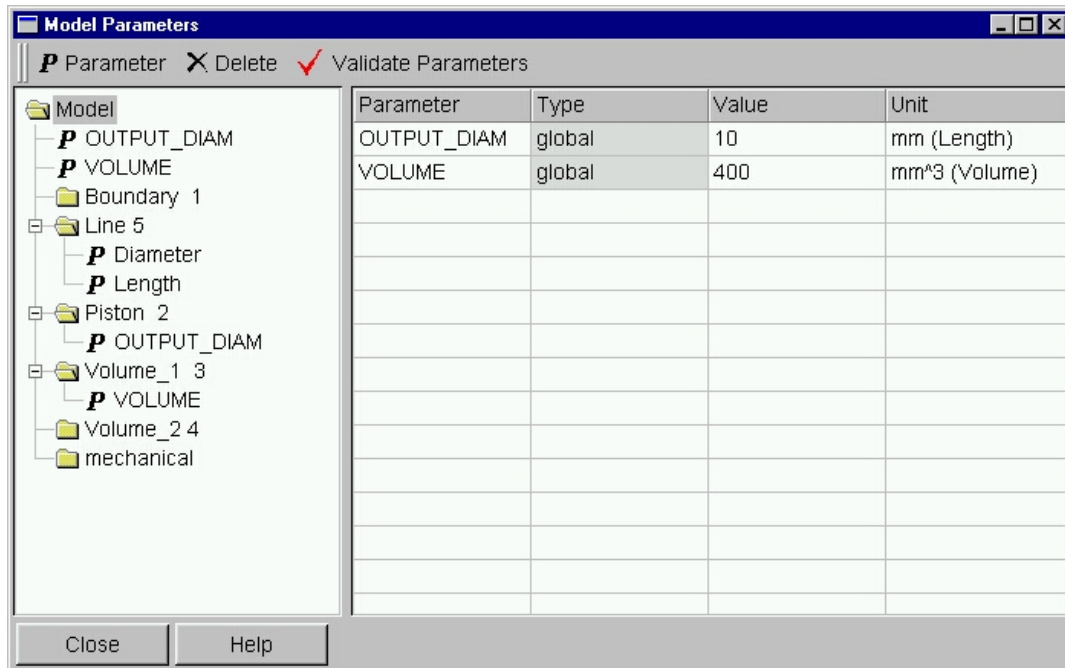


Figure 4-27: Model Parameters Window

In this dialog you can see the parameter tree with all existing parameters on the left side. It contains nodes for all elements of the model. All global parameters are displayed at the top of the tree under Model icon.

On the right side you can edit the parameter values. Constant values or expressions can be used. Click **Help** for more information.

4.7.4. Selecting Model Domain

The content of the table changes with the selection made in the parameter element tree: in *Figure 4-27* the model domain is selected, consequently the table contains all global parameters of the model. You can edit in the table all global parameters except those used for parameter variation by the Case Explorer, as will be explained later.

4.7.5. Selecting Element Domain

After selecting an element in the parameter tree, the table shows all parameters used by this element. For Line element the parameter list is shown in the following figure:

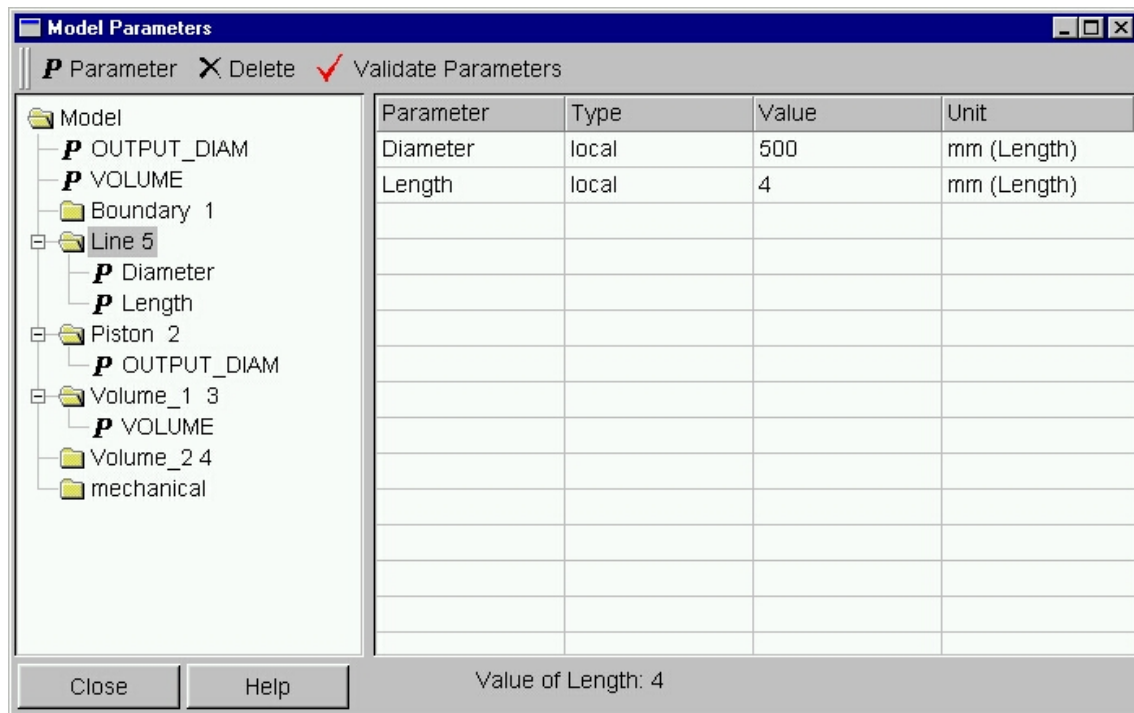


Figure 4-28: Parameters of the Selected Element (Line)


In this dialog you can edit all local parameters as well as change the type of parameter (global or local). To do this click in the Type field of the appropriate parameter and  button will pop up on the right end of the field. Click it and select another parameter type, as shown below:



Figure 4-29: Selecting Another Parameter Type

4.7.6. Using Global Parameters in Elements

An element accesses the global parameter with the help of another, element-specific parameter (e.g. parameter VOLUME for Volume1 element) of auto (global) type. As suggested by the name “auto (global)”, the element parameter value is inherited automatically from the global parameter with the same name.

4.7.7. Using Local Parameters in Elements

Local parameters do not need their global counterparts. Viewing and editing local parameters is feasible via the element Pop-up menu. Highlight the element and click the right mouse button. From the Pop-up menu select **Parameters** (refer to *Figure 4-30*).



Figure 4-30: Element Pop-up Menu





Note: Assigning new parameters can be performed in the Parameter Dialog box (**Model | Parameters**, refer to *Figure 4-27* and *Figure 4-28*). To add a new parameter, select either Model (for global parameter) or the appropriate element (for local parameter) and press **P** Parameter button. New parameter Parameter_1 will appear in the parameter tree and table. To delete the existing parameter, use **X** Delete button.

4.8. Case Explorer



Case Explorer is a tool for defining parameter variations (refer to *Figure 4-32*). To open the Case Explorer window, select **Model | Case Explorer**.


On the left side you find the case tree with a list of all model cases. Each case represents a simulation variant, i.e. a particular set of parameter values. The values of each case are specified in the case table on the right side.

Any number of Case Sets can be created in the Case Explorer Window by clicking the  button. Only a single Case within all Case Sets can be active at one time.

Delete Case Sets by selecting the appropriate Case Set and then clicking the  button. Case Sets can be renamed by clicking on the name in the case table and typing new text.


4.8.1. Add and Activate Cases

New model cases can be added by clicking the  button. Deleting cases can be performed by selecting the appropriate case and then clicking the  button.

The active model case in the model tree is marked with a red button. The set of parameter values defined for this case will be used when the simulation is started without an explicit case selection, e.g. with the Simulation icon  in the Palette menu. In addition, parameter dialogs will show values of this parameter set. To change the active case, double click the case in the case tree.

4.8.2. Add / Edit Parameters

Assigned case parameters (described in section 4.7) can be varied for the selected case set.

1. Select **Group | Edit** or  to add them to the case set(s). The following dialog appears:

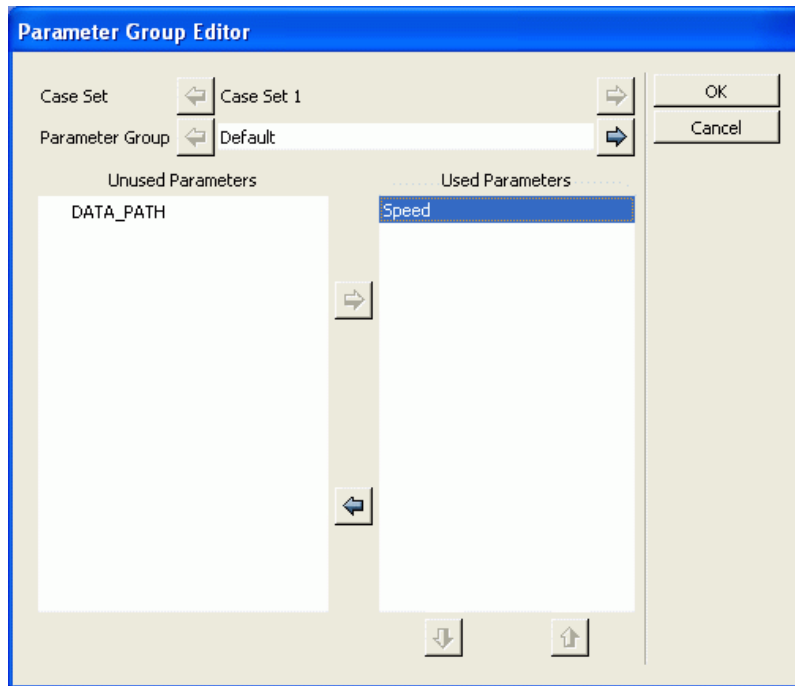




Figure 4-31: Parameter Group Editor Dialog

Use the left and right arrows for **Case Set** and **Parameter Group** to select different case sets and/or parameter groups.

Select the required **Unused Parameter** and click  to add it to the **Used Parameters** list. Select **OK**.

 shows parameters with constant values (no expressions).

2. In the Case Explorer table enter the relevant values for each case.

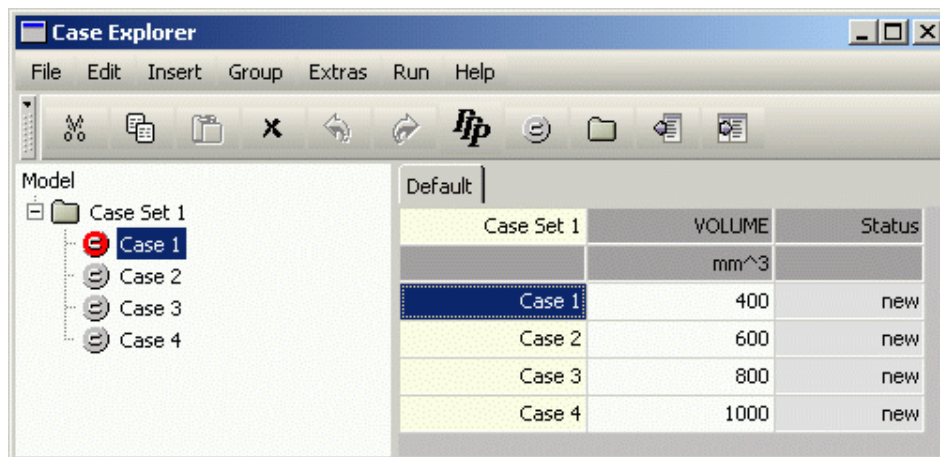


Figure 4-32: Case Explorer Window

Save these changes under a different file name, e.g. `simple_line_4cases.hyd`.

4.8.3. Run Simulations for Parameter Variations

For executing multiple cases select **Run** in **Simulation** menu. The following dialog will appear:

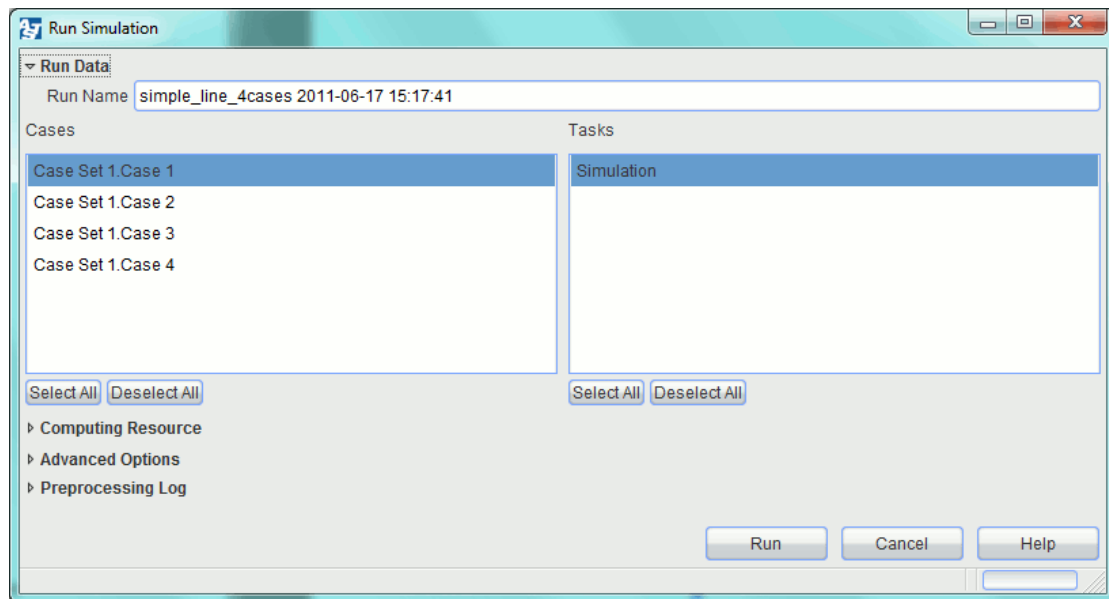


Figure 4-33: Run Dialog with Multiple Cases

Select the desired cases in the case list. To extend the selection hold the SHIFT or CTRL button pressed while clicking case names. When ready press **Run**. Simulation jobs are pre-processed and submitted. The Workspace user interface will pop up a Simulation status window showing information on job submission.

Refer to the Online [Help](#) for more information on job submission.

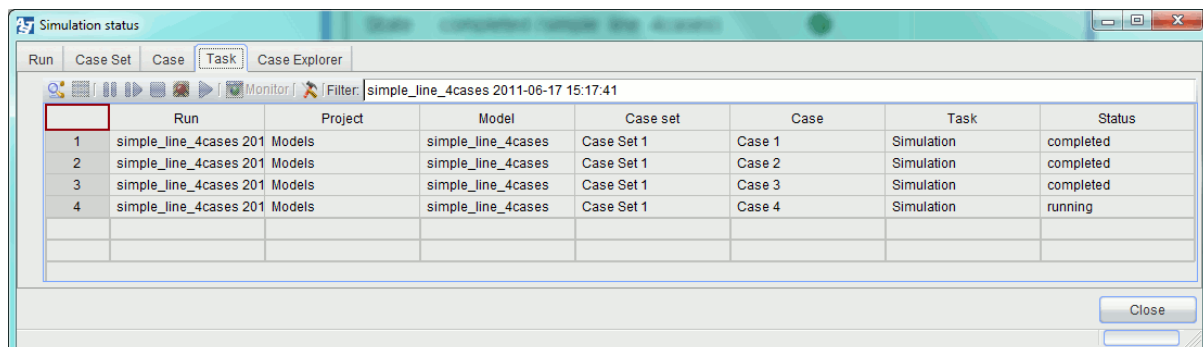


Figure 4-34: Simulation status with running Case 4

Calculation results will be stored in case-specific sub-directories as follows:

`<project_dir>/<model_name>.<case_set>.<case_name>/results.ppd`

Refer to *Section 4.5* to view calculation results.

4.9. Thermal Calculation

To switch between the isothermal and thermal fluid flow calculation, open **Simulation | Mode** menu and select the desired **Calculation Task**.

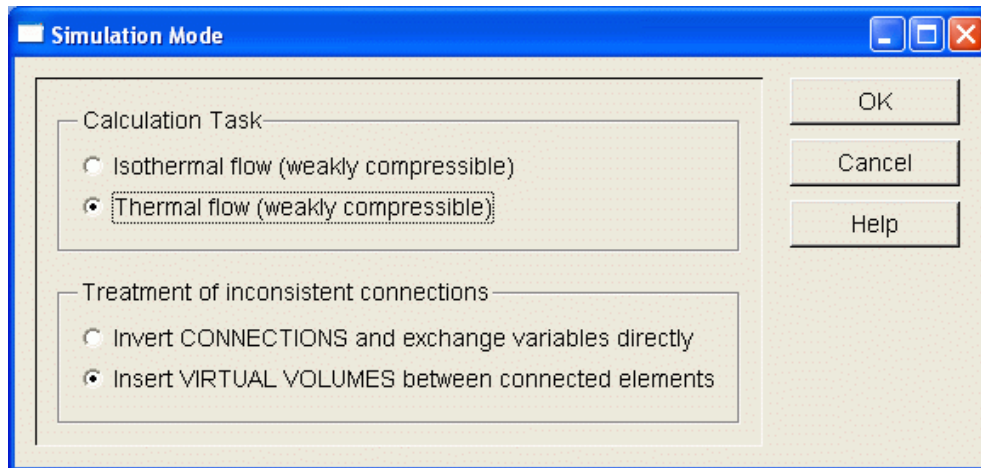


Figure 4-35: Simulation Mode dialog

For fluid flow, rigorous thermal calculation is fully implemented in Volume group and first two Orifice group elements only. Gas group elements are always calculated in thermal mode (irrespective of the above selection). All other elements use averaged temperature for the interpolation of local fluid, gas and solid properties. The temperature of hydraulic flow elements (lines, orifice, valves) is determined from the connected Volume and Boundary elements.

Open existing example `simple_line1_lp.hyd` (with Laplace line model). Activate **Thermal flow** in **Simulation | Mode** dialog.

To plot Volume temperature and variable fluid properties (bulk modulus, density), add these parameters in the Volume Output Parameters dialog box.

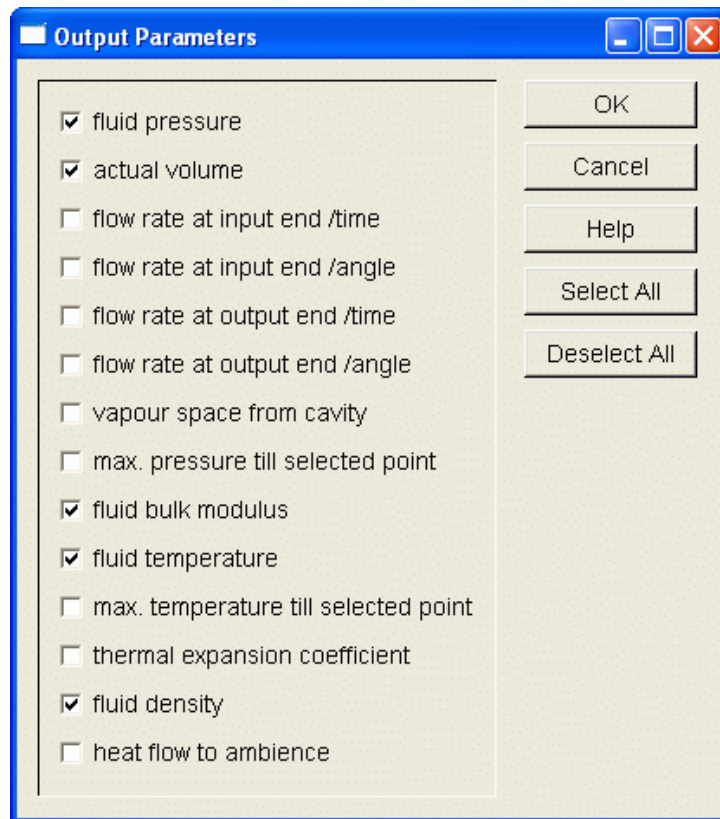


Figure 4-36: Volume output parameters

Set initial temperature of Volume1 element to 50 degC, and Volume2 - to 20 degC. Next, set the thermal expansion coefficient in **Global Fluid Properties** dialog to 0.01 1/degC.

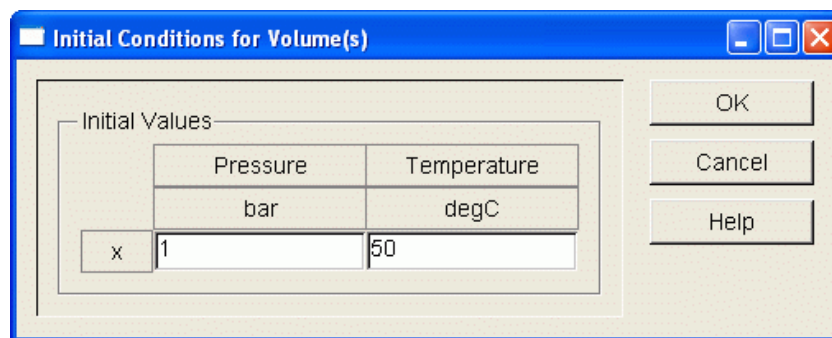


Figure 4-37: Volume initial conditions

Save these changes under a different file name, e.g. `simple_line1_lp_thermal.hyd`. Extend calculation end time from 0.01 s to 0.03 s in **Calculation Control** dialog and restart the calculation. Longer calculation period should be used because the fluid temperature is changing much more slowly compared to the pressure.

4.9.1. Thermal Calculation Results

In order to compare results between isothermal and thermal calculation, load results of the models `simple_line1_lp.hyd` and `simple_line1_lp_thermal.hyd` model into Impress Chart. In **Result** tree, for both models load pressure in Volume2 into the layer 1, temperature from Volume1 and Volume2 into layer 2 and fluid density into layer 3. Adjust the layer title and legend text. The result comparison is shown in *Figure 4-38*.

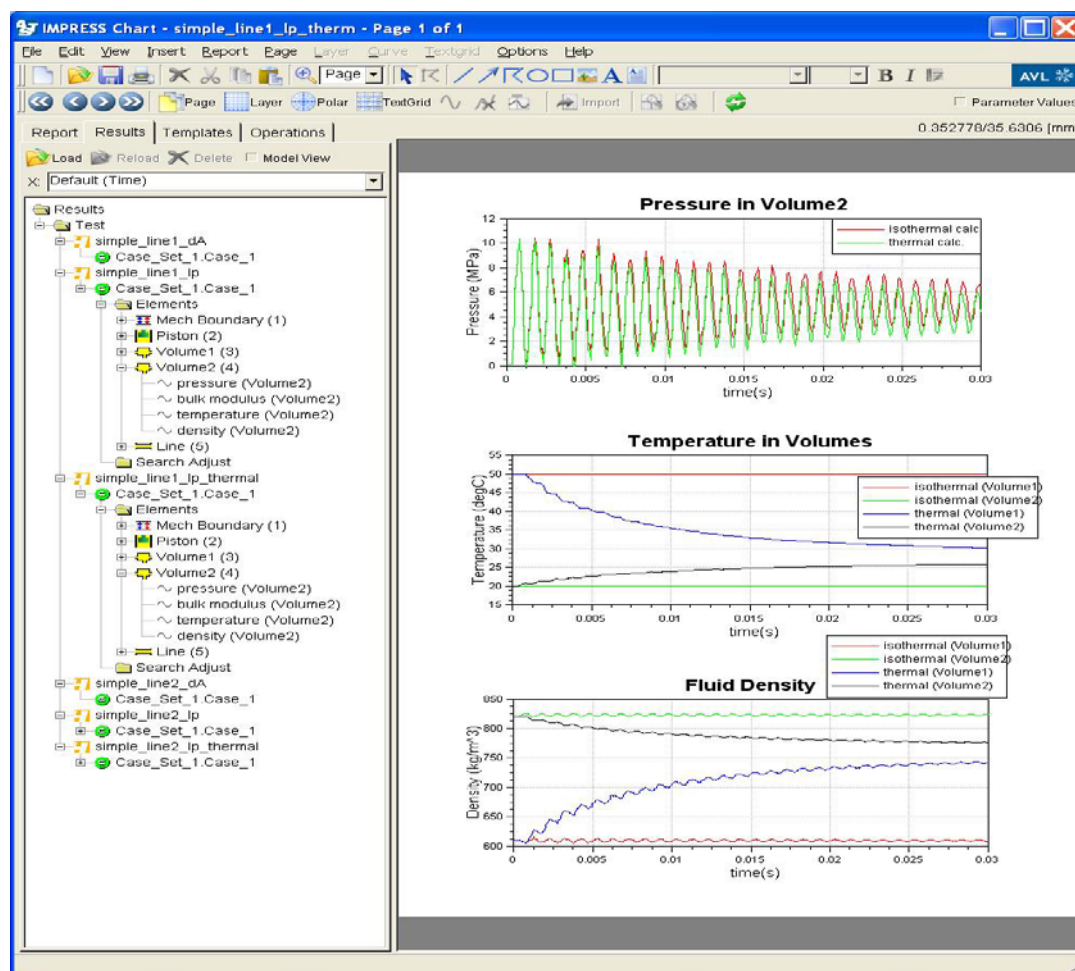


Figure 4-38: Result comparison between isothermal and thermal calculation

