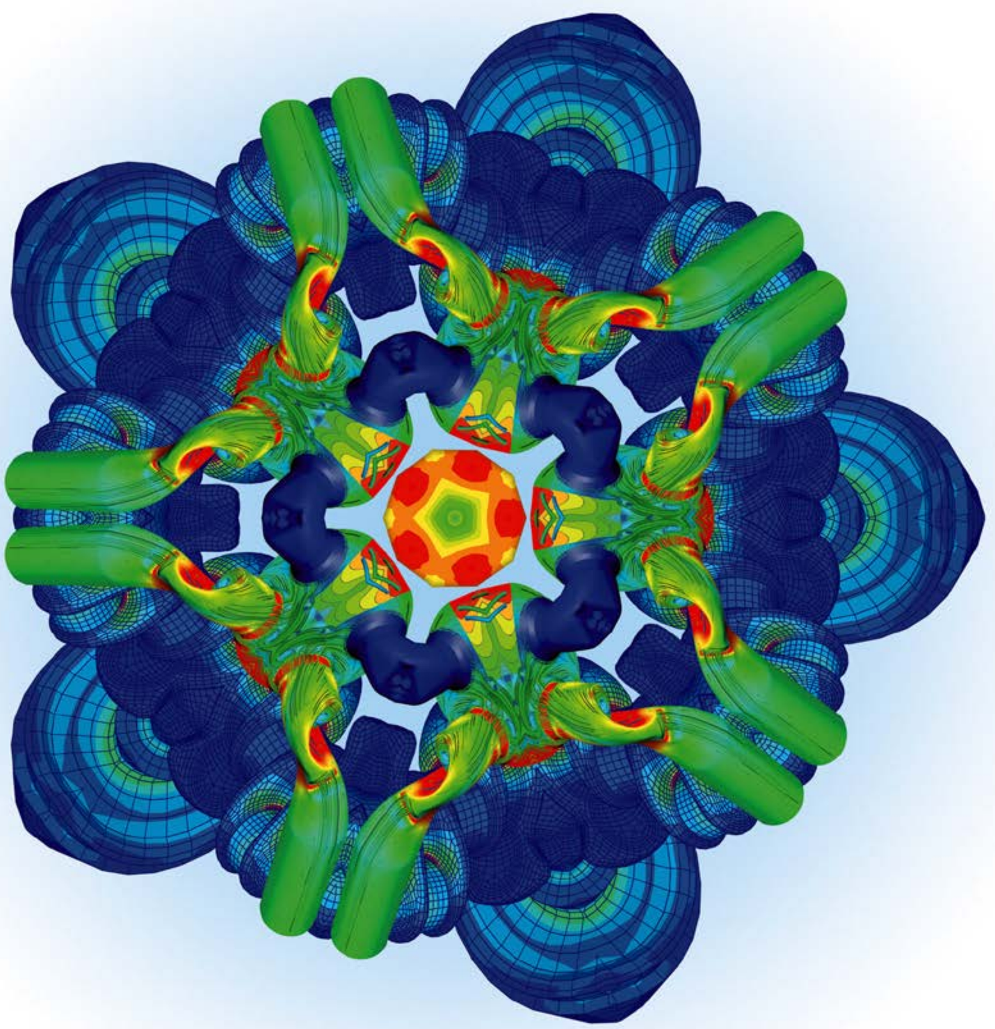
Edition 04/2013

Hydsim Primer

AVL BOOST
VERSION 2013.1



AVL LIST GmbH
Hans-List-Platz 1, A-8020 Graz, Austria
http://www.avl.com

AST Local Support Contact: www.avl.com/ast-worldwide

Revision	Date	Description	Document No.
A	30-Apr-2009	HYDSIM v2009 - Primer	11.0102.2009
B	17-Jul-2009	HYDSIM v2009.1 - Primer	11.0102.2009.1
C	30-Mar-2010	HYDSIM v2009.3 - Primer	11.0102.2009.3
D	19-Nov-2010	BOOST Hydsim v2010 - Primer	11.0102.2010
E	29-Jul-2011	BOOST Hydsim v2011 - Primer	11.0102.2011
F	30-Apr-2013	BOOST Hydsim v2013.1 - Primer	11.0102.2013.1

This document describes how to run the BOOST Hydsim software on your computer. It does not attempt to discuss all of the concepts of fluid mechanics and multi-body dynamics that are required to obtain successful solutions. It is your responsibility to determine if you have sufficient knowledge and understanding of fluid mechanics and dynamics to apply this software appropriately.

Table of Contents

List of Figures

1. INTRODUCTION

This document describes the basic concepts and methods for using BOOST Hydsim v2013.1 to calculate the dynamics in hydraulic and hydro-mechanical systems.

1.1. Scope

The chapters of this manual describe how to run the BOOST Hydsim software. They do not attempt to discuss all concepts of fluid mechanics and dynamics that are required to obtain successful solutions. It is the user's responsibility to determine if he/she has sufficient knowledge and understanding of the dynamics of fluids and solids to apply this software appropriately.

1.2. User Qualifications

Users of this manual:

- Must be qualified in basic Linux or Windows
- Must be qualified in basic Fluid Mechanics and Multi-body Dynamics
- The Primer is a basic qualification for operating BOOST Hydsim and users are recommended to continue with BOOST Hydsim training courses and to refer to the BOOST Hydsim Users Guide for further in-depth information.

1.3. Symbols

The following symbols are used throughout this manual. Safety warnings must be strictly observed during operation and service of the system or its components.

⚠ **Caution:** Cautions describe conditions, practices or procedures which could result in damage or destruction of data if not strictly observed or remedied.

☞ **Note:** Notes provide important supplementary information.

Convention	Meaning
Italics	For emphasis, to introduce a new term.
`monospace`	To indicate a command, a program or a file name, messages, or input / output on a screen or file contents.
MenuOpt	A **MenuOpt** font is used for the names of menu options, submenus and screen buttons.

1.4. Configurations

Software configurations described in this manual were in effect on the publication date of this manual. It is the user's responsibility to verify the configuration of the equipment before applying procedures in this manual.

2. OVERVIEW

BOOST Hydsim is a program for the dynamic analysis of hydraulic and hydro-mechanical systems. It is based on the theory of fluid dynamics and vibration of multi-body systems. The main application area of BOOST Hydsim is the simulation of fuel injection. Historically, the program has been developed for the simulation of diesel injection systems. Today BOOST Hydsim is well suited for the modeling of gasoline, heavy oil, alternative fuel injection etc. Moreover, new challenging applications such as hydraulic transmissions, valve trains and actuators have been covered. BOOST Hydsim can be used for the multi-cycle simulation of the injection and other systems including control units. Generally, the program is useful in a variety of fields concerned with dynamic analysis of hydraulic, mechanical and control systems. For instance, using MATLAB® interface, BOOST Hydsim can simulate the dynamics of hydro-mechanical control devices, such as for engine inlet and exhaust valve lift control, and the vibration of drives.

BOOST Hydsim is an integrated tool of the AVL Workspace with a user-friendly graphical pre- and post-processing. The 2-dimensional (2D) representation of the BOOST Hydsim model provides a general image of the system as defined by the user. Basically, each particular element of the physical system is represented by an icon (symbol containing schematic figure of the physical element) on the GUI (Graphical User Interface) screen. The icons can be mechanically, hydraulically or logically connected. GUI controls the model build-up process and does not allow incompatible connections or other invalid input specifications.

2.1. Model Structure

A system model using BOOST Hydsim normally contains various hydraulic, mechanical and control elements. Elements are joined into groups according to their type and functionality and are listed in the Elements Tree. Refer to Section 3.3 for description.

2.2. Main Input Data

The input data depends on the system configuration and the task specified - a standard run, restart, run with optimization or series calculation. A fixed set of input parameters is associated with each element. Some parameters are optional (supplied with switches). Each element is characterized by an ID number and user-defined name. Fluid properties and mechanical connections require separate input. Furthermore, general data for calculation control have to be specified.

2.3. Results

Each element has a predefined set of results which (if selected by the user), are stored on diverse ASCII files. By default, the data and control information are stored on the GIDas file. Its contents can be directly opened with the Case Explorer, which is integrated in the Impress Chart post-processor. For optimization run, an iteration history file is also produced (GAD File). Output of results is available in time domain (always) and domain of shaft rotation angle, if selected.

Typical simulation results for hydraulic elements are:

- Pressure
- Temperature
- Flow rate (volumetric rate or mass rate)
- Cumulative rate (volumetric or mass)
- Geometric and effective flow area
- Flow resistance/discharge
- Vapor cavity
- Cavitation factors

For mechanical elements, typical simulation results are as follows:

- Coordinate, velocity and/or acceleration
- Dynamic forces and torque
- Kinematic parameters

The post-processing of the data (plots) is carried out by the interactive 2D post-processor (Impress Chart) of the AVL Workspace. Impress Chart allows a flexible automated generation of plots by using predefined templates (delivered with BOOST Hydsim or designed by the user), as well as the interactive creation of graphs, diagrams, etc.

2.4. AVL BOOST Hydsim Pre-processor

The pre-processor/GUI enables the user to:

- Build a 2-D representation of the BOOST Hydsim model
- Define properties and other specifications
- Generate boundary excitation data
- Perform calculation (single or multiple runs)
- Access Impress Chart for result evaluation
- Access PP3 for Injector/Nozzle Flow animation

2.4.1. 2-D Representation

The purpose of the 2-D representation of the BOOST Hydsim model is to provide a general image of the system as defined by the user. Basically, an icon (symbol) on the GUI screen represents each particular element of the physical system. Icons contain schematic figures of the physical elements. Icons are connected by red or blue lines with arrows. A red line implies a mechanical connection (spring and/or damper) and a blue line – a hydraulic connection (flow direction). Certain elements may be connected by green lines (special connections) as discussed in *Section 3.4.3* and by orange lines (wire connections) as discussed in *Section 3.4.4*.

2.4.2. Define Properties

Once the 2-D representation of the system is defined, the properties of the elements and mechanical connections may be specified. For this, select the icon with the left mouse button and click the right button to open the input dialog box of the element. In addition, by opening different dialog boxes from the Menu bar, the user may specify the initial conditions, desired output parameters, and define other properties related to the element. Properties of mechanical connections (red lines) are specified in the same way. Hydraulic (blue), special (green) and wire (orange) connections have no user-defined properties.

2.4.3. Generate Boundary Excitation Data

All external excitation in BOOST Hydsim is specified through the Boundary Condition elements (hydraulic-pressure or flow rate, mechanical or hydromechanical). The Mechanical Boundary is shown in the model in *Figure 4-3*. Basically, both mechanical and hydraulic excitation can be specified in the form of displacement, velocity, pressure and flow functions of time or shaft rotation angle.

2.4.4. BOOST Hydsim Execution

Run the BOOST Hydsim program directly from the GUI by clicking **Run** in the **Simulation** pull-down menu. Ordinary Run or Restart procedure will be executed.

☞ **Note:** BOOST Hydsim execution can be started only if all necessary data are formally supplied. In addition, a series of data compatibility checks is performed by GUI before the calculation start. If calculation cannot be started an appropriate error/warning message will be issued on the screen.

If the GUI main screen or operating system has not issued any error message, BOOST Hydsim has started the execution of the model. During numerical calculations various information, warning and error messages may be produced by the internal calculation routines. These are stored on a text file `simulation.out` and can be viewed by selecting **View Logfile** in the **Simulation** menu. It is ultimately recommended to use this option after each start of the program (especially with a new model). Any program break-up caused by a fatal run-time error or data compatibility violation will be reported there.

2.4.5. IMPRESS Chart Post-Processor

The Impress Chart post-processor may be accessed directly from the GUI to view 2-D plots of results. The Impress Chart tool is used for the result evaluation of the BOOST Hydsim simulation. By default results are plotted as a function of time and, if appropriate, shaft rotation angle. However, any output parameter can be selected as output domain (x axis). The desired output parameters must be selected from a predefined list in the GUI preprocessor. For injection systems, the typical output parameters would be needle and piston motions, injection rates and quantities, flow areas, pressures in injection line (nozzle and pump side), pump and nozzle chambers, etc. The Impress Chart Case Explorer gives a direct access to the output data (including optimization results) within an easy-to-use, self-explanatory layout.

Refer to the Impress Chart section of the GUI Users Guide for more information.

2.4.6. PP3 Post-Processor

The PP3 post-processor may be accessed directly from the GUI to perform animation of injector nozzle flow. PP3 animation is available only for BOOST Hydsim models containing Basic or Extended Nozzle Orifice elements (SAC or VCO type). Extended animation features are available for a standard common rail injector with 2/2-way control valve and unit injector with split-injection device (SID).

Refer to the *PP3* section of the GUI Users Guide for more information.

3. GETTING STARTED

This chapter describes the tasks and concepts needed in order to start and use BOOST Hydsim.

3.1. Starting AVL Workspace (AWS)

3.1.1. Windows

Open the AVL Workspace from the **Start** / **Programs** menu or double click the icon on the desktop. Click on the BOOST Hydsim logo or select BOOST Hydsim from the Programs menu to open the main window as shown in Figure 3-1.

3.1.2. Linux

Open the AVL Workspace by executing the command from the Linux or DOS prompt

```
% aws
```

in your working directory.

3.2. Starting a BOOST Hydsim Session

Once the **Workspace** menu is opened, select BOOST Hydsim to begin a new session of the program.

BOOST Hydsim can be started by clicking BOOST Hydsim directly in the window.

A BOOST Hydsim session can be directly started from the Linux or DOS prompt by the following command:

```
% aws BOOST Hydsim
```

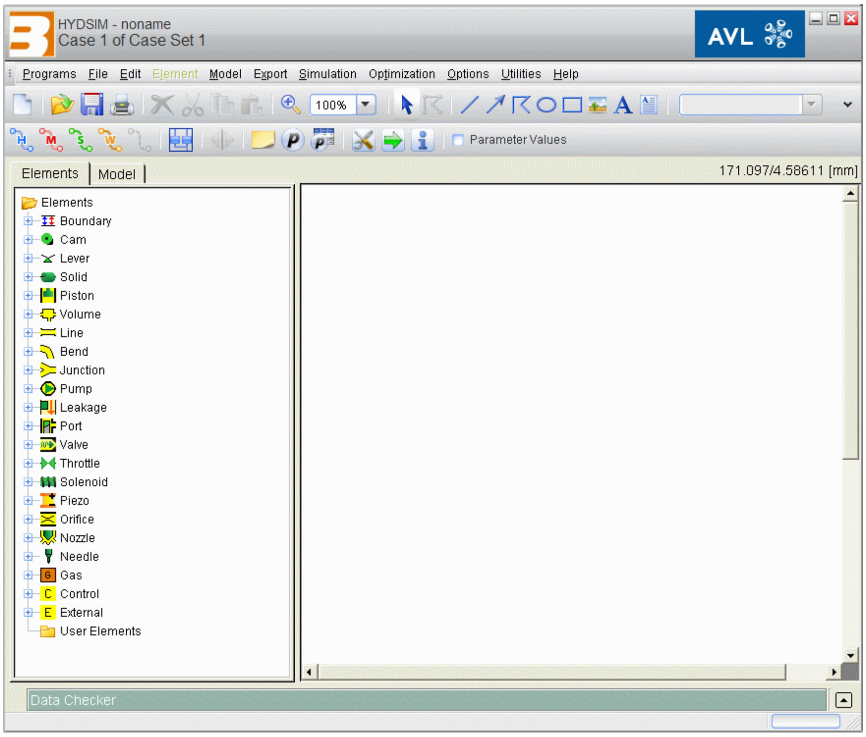


Figure 3-1: BOOST Hydsim Main Window

Menu	Submenu	Description
Programs	**New**	Opens a new session of BOOST Hydsim.
	Exit	Terminates a Workspace session.
File	**New**	Opens a new session of the Workspace (clears the current session).
	Open	Opens a saved session of the Workspace.
	Save	Saves the current session of the Workspace.
	Save As	Saves the current session of the Workspace with a different name.
	Page Setup	**Size:** Check either **Portrait** or **Landscape** for **Orientation** of the page and define **Size** in percent of a standard A4 page. **Grid:** Check **Grid active** to switch on the grid. Checking **Grid visible** shows the grid on the page. **Grid Size** defines the vertical and horizontal distance between grid-points. **Offset** defines the start of grid from upper left corner of the page. **Unit:** Set the unit option **mm, cm, inch** or **pt** (points).
	Print	Saves the current session of the BOOST Hydsim Workspace to a postscript file for printing.

Edit	**Cut, Copy, Paste, Delete**	Cut, copy, paste and delete commands for the creation and manipulation of the elements.
	Modules	Modules are used to build a database system where different element combinations in different modules can be stored. These can be put together into one model later. With this type of modeling, input work can be minimized and checking of part systems can be simplified. **Load Module:** Select **Load Module** to open a file selection dialog where any previously saved module can be inserted into the actual model by selecting the module file (default-extension .mod). **Save Module:** Parts of models can be saved as modules. First select the part of the model in the Working Area. A file selection window opens to specify a file name to save the module (default extension .mod). Modules saved with **Save Module** are not introduced into the element tree.
		Save As User Module: Proceed as with **Save Module**. The directory where user modules are saved is predefined as (`AWS_USERHOME/clients/tycon/lib/macros` – where AWS_USERHOME is an environment variable). After specifying the file name (default extension .mod), select enter to confirm. The Saving Module window opens and the user selects a branch of the element tree where the module will be included. All users have authorization to define user modules.
		Save As Site Module: Proceed as with **Save Module**. The directory where the site modules are saved is predefined as (`AWS_SITEHOME/clients/tycon/lib/macros` – where **`AWS_SITEHOME`** is an environment variable). After specifying the file name (default extension `.mod`), select **ENTER** to confirm. The **SAVING MODULE** window opens and the user selects a branch of the element tree where the module will be included. Only users with authorization to write on `AWS_SITEHOME` can define site modules (due to file protection).
	Blocks	**Create:** Select or drag the mouse over the required elements to group together. **Break up:** Ungroups the elements of the block.
	Select	**All:** Selects everything in the Working Area. **All Elements:** Selects all elements from the Element Tree in the Working Area. **All Connections:** Selects all element connections used in the model. **All Decorations:** Selects all graphical elements on the screen, e.g. rectangle, circle, EPS, Text...

	Order	**Raise:** If elements overlap fully or partly, bring an element to the foreground by selecting it. **Lower:** The selected element is sent to the background.
	Group	**Group:** Connects several selected graphical elements together. Surround the graphical elements with a rectangle using the mouse pointer. Hold the left mouse button and click Group. The selected elements are permanently grouped as required. **Ungroup:** Cancels the grouping of graphical elements.
Element	**Properties…**	Specify properties of the selected element
	Initial Conditions	Specify initial conditions of the selected element
	Store Results…	Specify output parameters for the selected element
	Modify…	Specify modifiable parameters for the selected element
	Copying…	Copy element properties from one element to other element(s)
Model	**Parameters…**	Edit model parameters
	Case Explorer…	Define parameter variations.
	Fluid Properties…	Define global fluid properties.
	Solid properties…	Define global solid properties.
	Cylinder Charge…	Define cylinder charge properties.
	Property Database…	Define fluid properties in Property Database.
Export	**FIRE Flow/ Spray**	Define data for link to FIRE
Simulation	**Run…**	Submit simulation jobs for one or several parameter sets (cases).
	Restart…	Restart (further calculation) of previously saved system.
	Control	Define the solution control of the simulation. Time steps can be defined, simulation intervals and result intervals.
	Mode…	Defines calculation task (isothermal and thermal) as well as treatment of inconsistent connection (with or without virtual Volumes).
	Search Adjust…	Defines parameters for optimization.
	Animation | Nozzle Flow …	Opens dialog box either to define animation parameters or call PP3 to show a 3D animation of the calculated model.
	Status…	Shows simulation status.
	View Logfile…	Shows information, warning and error messages from the calculation routines.

	Show Results	Opens the Impress Chart window with the right model directory.
Options	**Computing Resources**	Define job submission settings: queues, number of processors per job and number of parallel jobs.
	Model Locking	Lock the property dialogs for elements for simplified and protected model views.
	Frame	Set of graphical elements used for page layout, e.g. rectangle (frame), logo and text elements. **None:** Removes the frame from the page. **AVL Report:** The standard AVL frame in portrait. **AVL Report Landscape:** The standard AVL frame in landscape. **Customer Report Landscape:** Customer's frame in landscape.
	Frame Definitions	Customized settings of the current frame. Specify text and the customer logo for the frame.
	Units	Used to display and set the units used. Refer to the GUI Users Guide, Section 2.4.2.
Help	**Contents**	Online Help.
	Manuals	Access BOOST Hydsim manuals in PDF format.
	About	Brief information on the current BOOST Hydsim release.

3.2.1. Programs

This menu is the gateway to the other suite of AVL engine simulation programs. The first selection, **New**, opens a new session of any of the programs without having to tie up more CPU memory. This is also where the current session of the Workspace can be exited. By selecting **Exit BOOST Hydsim**, the session of BOOST Hydsim will also be terminated.

> **Note:** If you have only a BOOST Hydsim license, you can work only with three programs: BOOST Hydsim and postprocessors Impress Chart and PP3. However, viewing of other AWS software codes is possible.

3.2.2. File

This menu allows the user to do the following:

- **New**  : Clears the current BOOST Hydsim Workspace and starts a new session
- **Open**  : Opens a saved session of the BOOST Hydsim Workspace
- **Save**  : Saves the current session of the BOOST Hydsim Workspace
- **Save As:** Saves the current session of the BOOST Hydsim Workspace under a different name

- **Print**  **:** Sends the current session of the BOOST Hydsim Workspace window to a printer or saves on postscript file
- **Exit BOOST HYDSIM:** Exits the current BOOST Hydsim Workspace and prompts the user for verification

3.2.3. Edit

This menu allows the user to perform standard screen editor commands (**Cut, Copy, Paste, Delete, Select,** and **Order**) for the creation and manipulation of the BOOST Hydsim workspace model. In the command **Modules**, **Load Module**, **Save Module** and **Save as User/Site/System Module** features are available. **Save Module** saves a selected part of the model on file with default extension `*.mod`. **Load module** allows reading of an earlier saved part of the model from an existing `*.mod` file. The **Select** option allows the user to select the GUI objects by different criteria. **Order** option serves to Raise or Lower the selected icon on the screen. **Delete, Cut, Copy**, and **Paste** functions can be directly performed by clicking the respective buttons: , , , .

Menu item **Initial Conditions** allows automatic definition of initial conditions for element groups. Before specifying initial conditions, the desired elements must be selected either with **Select** menu, or directly with the mouse in the BOOST Hydsim window. With **Initial Conditions**, the initial values must be entered in one step for the appropriate elements of the entire model or its part.

Under **Blocks** item are available features, which allow creating blocks, i.e. container objects for a number of elements and connections. The elements and connections in the block cannot be destroyed and are not accessible for graphical interactions such as moving or sizing. This makes a block the right thing for grouping editor elements. By the same token a block can also be used as a completely reusable component that stores the connection information so that it can be attached to other elements quickly.

3.2.4. Simulation

This menu controls the BOOST Hydsim simulation. The data for numerical integration and global fluid properties have to be supplied here. The menu contains the following:

- **Run**: Ordinary run of single or multiple cases
- **Restart**: Restart (further calculation) of previously saved system
- **Control:** Defines calculation domain, numerical data for simulation, and output
- **Mode:** Defines calculation task (isothermal and thermal) as well as treatment of inconsistent connection (with or without virtual Volumes)
- **Search Adjust:** Defines parameters for optimization
- **Animation | Nozzle Flow :** Shows animation of Nozzle flow
- **Status :** Shows simulation status
- **View Logfile:** Shows information, warning and error messages from the calculation routines
- **Show Results:** Opens the IMPRESS Chart window with the right model directory

Refer to BOOST Hydsim Users Guide for more information.

3.2.5. Other Menus

Element details will be described later in this manual. Other menus can be easily viewed by the user and are self-explanatory.

3.3. Element Library

The system model in BOOST Hydsim may consist of various hydraulic, mechanical and other elements. Elements are collected into groups according to their type and functionality. BOOST Hydsim has nineteen element groups whose names are listed in the Elements Explorer.

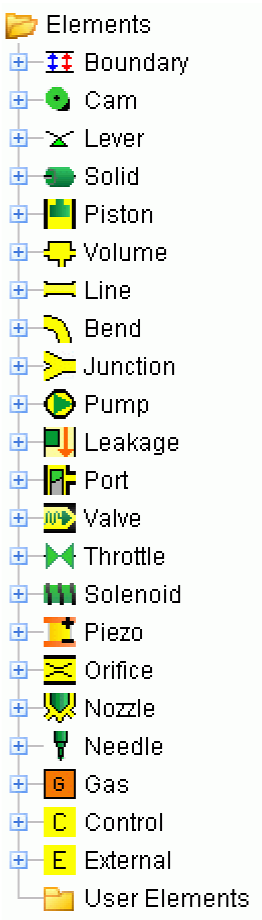
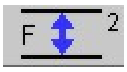
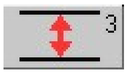
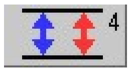
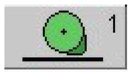
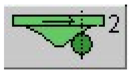
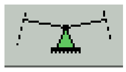
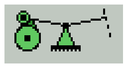
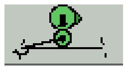
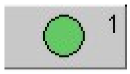
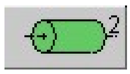
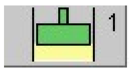
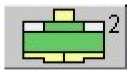
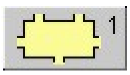
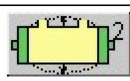


Figure 3-2: Element Group Tree

Double-click on the group with the left mouse button to show the entire list of elements belonging to that group. The complete group and their element list as well as element icons are tabulated below (*Table 3-1*). Refer to the BOOST Hydsim Users Guide for more information.

Table 3-1: Element Library

GROUP	ELEMENT	ICON
 **Boundary**	**Pressure** Defines the pressure on outside connections (boundaries) of the system as function of time or reference angle.	
	Flow Rate Defines the flow rate on outside connections (boundaries) of the system as function of time or reference angle.	
	Mechanical Defines the coordinate or velocity on outside connections (boundaries) of the system as function of time or reference angle.	
	Hydromechanical Defines the coordinate/velocity and pressure and/or flow rate on outside connections (hydraulic and mechanical boundaries) of the system as functions of time or reference angle.	
 **Cam**	**Cam Profile** Defines a cam profile by the follower acceleration or lift data (grinding coordinates for reciprocating follower).	
	Cam Plate * Defines a cam plate of distributor-type injection pump (e.g. Bosch VE30) by acceleration of lift data.	
 **Lever**	**Rocker Arm (Pushrod)** * Defines a pushrod-actuated rocker arm.	
	Rocker Arm (Cam) * Defines a cam-actuated rocker arm. It includes cam profile in polar coordinates.	
	Finger Follower * Defines a finger fo*llower (semi-floating rocker). It includes cam profile in polar coordinates.	
 **Solid**	**Mass** Defines a lumped mass (with two DOFs).	
	Shaft Defines a rigid or elastic shaft (rigid - with three DOFs).	
 **Piston**	**Standard** Defines a standard (hydraulic) piston.	
	Split-Injection (SID) * Defines a split-injection-device piston. SID piston is used for rate shaping of fuel injection (e.g. in pilot injection systems).	
 **Volume**	**Standard** Defines a standard volume with rigid walls.	
	Compliant Defines a volume with compliant walls (cylindrical or spherical).	

* At this moment Variable-step solver does not support these elements.

	Two-phase * Defines a two-phase volume (fluid-gas mixture).
Line	**d'Alembert Model** Defines a line (duct/tube/pipe). The solution of the line equation (without frictional losses) is derived by d'Alembert. Friction function is defined through empirical pressure-pulse damping.
	Laplace Transform Defines a (duct/tube/pipe) line of circular form. The solution of the line equation is obtained by Laplace transform (Kroller[1]). The non-stationary friction losses are calculated by Melcher's method.
	Characteristics Method * Defines a line (duct/tube/pipe) solved by the method of characteristics. For integration, the **Predictor - Corrector scheme** is used. The non-stationary frictional losses are calculated by Melcher's method.
	Godunov Method * Defines a line (duct/tube/pipe) solved by the Godunov method. The frictional losses can be calculated by Melcher's method, Colebrook-White equation or using a fixed friction factor.
	MacCormack (Two-phase) Defines a line (duct/tube/pipe) solved by MacCormack finite difference scheme[2]. For two-phase flow model, the bubble dynamics theory is applied. MacCormack scheme is combined with FCT (Flux Corrected Transport) algorithm. The frictional losses can be calculated by Melcher's method, Colebrook-White equation or fixed friction factor.
Bend	**Round/Circular** * Defines a round bend of circular cross-section and constant area.
Junction	**Tee (90 deg)** * This element serves to define a Tee junction with 90 deg angle between side and main branch.
	Tee (angle) * This element serves to define a Tee junction with arbitrary angle between side and main branch.
Pump	**Radial Piston (RPD)** * Defines a radial piston distributor (RPD) pump. The model can account for the elastic Hertz contact between the pump elements.
	Plunger Defines the plunger (of any type).

[1] M.Kroller, *Efficient Computation of a Mathematical Model for the Damping of Pressure Waves in Tubes of Circular Form*. Numerical Methods for Partial Differential Equations, John Wiley & Sons, Inc., 1995.

[2] G. Regner and A. Hariyanto, *A Bubble-Dynamic Cavitation Model for the Simulation of Diesel Fuel Injection Systems.* MTZ *worldwide*, 61(2000) 7/8.

Leakage	**Annular Gap** Models the fluid leakage through the annular gap between piston-type element (Piston, Plunger, Needle guide, etc.) and barrel (cylinder wall).	
Port	**In-line Fill/Spill** Defines a round filling, spill, or fill/spill port of in-line pump.	
	Distributor Fill/Spill * Defines the filling/spill ports of distributor-type rotary pump (e.g. Bosch VE).	
Valve	**Delivery** Defines a delivery valve. It can be used standalone or as a part of snubber valve or constant pressure valve.	
	Constant Volume * Defines a constant volume delivery valve with a retraction piston.	
	Check (Ball) Defines a pressure check valve with spherical body (ball).	
	Check (Poppet) Defines a pressure check valve with conical body.	
Throttle	**Time-controlled (Switch)** Defines a switch valve controlled by the timing of opening/closing areas (flow area as function of time or crank angle).	
	Lift-controlled (Slide) Defines a slide (spool) valve controlled by position of a mechanical body (flow area as function of position).	
	Flow Area vs. Time/CA Defines a throttle controlled by flow area variation vs. time (area as function of time or crank angle).	
	Pressure Drop vs. Flow * Defines a throttle controlled by pressure-flow diagram (pressure difference as function of flow rate).	
Solenoid	**Armature (Basic model)** Defines a solenoid armature actuated by a gap-dependent magnetic force.	
	Armature (Extended model) Defines a solenoid armature actuated by a variable magnetic force. (internal or external). Magnetic force is a 2D function of armature lift and amperage.	
Piezo	**Lift Function *** Defines a lift function of a piezoelectric actuator. It is treated as a boundary condition with a predefined charging/discharging function. The boundary function defines the motion of piezoelectric stack end.	
	Lift Amplifier * Defines a lift amplifier (force–lift converter). It converts the motion of two connected pistons into the forces acting on these pistons. It is a 2D force-displacement function.	

	Stack Actuator * Defines a piezoelectric stack actuator. It is a lumped-parameter model with rate-independent hysteresis describing the accurate actuator behavior in electrical and mechanical domains.	3
Orifice	**General** Defines a general orifice. It is not restricted to any specific geometry. Flow resistance coefficient has to be defined by the user.	1
	Cavitating Defines a cavitating orifice (flow model with cavitation). It is not restricted to any specific geometry.	2
	Sharp-edged * Defines a sharp-edged orifice. Flow resistance coefficient is calculated internally.	3
	Round-edged * Defines a round-edged orifice. Flow resistance coefficient is calculated internally.	4
	Sharp-edged Long * Defines a sharp-edged long orifice. Flow resistance coefficient is calculated internally.	5
Nozzle	**SAC (Basic model)** Defines a sac nozzle orifice with hydraulic flow (basic model).	1
	VCO (Basic model) Defines a valve-covered nozzle orifice with hydraulic flow (basic model).	2
	SAC (Extended model) Defines a sac nozzle orifice (extended model with cavitation).	3
	VCO (Extended model) Defines a valve-covered nozzle orifice (extended model with cavitation).	4
	RSN-Collar Throttle * Defines a needle collar throttle of rate-shaping nozzle (RSN).	5
	Non-circular Guide Throttle * Defines the flow properties through non-circular needle guide.	6
Needle	**Standard** Defines an up-to-date needle model of standard nozzle.	3
	2-spring Defines an up-to-date needle model of two-spring Injector (TSI) nozzle.	4

Gas	**Pressure/Temp Boundary *** Defines the gas pressure and temperature on the outer connections (boundaries).	
	Volume * Defines the properties of gas volume.	
	Lift-controlled Throttle * Defines a gas throttle controlled by a position of one or two mechanical bodies.	
	Flow Area vs. Time/CA * Defines a gas throttle with flow area as a function of time or crank angle.	
Control	**MATLAB® API: Simulink *** Defines an interface between BOOST Hydsim and MATLAB® API Simulink.	
	MATLAB® API: m-function * Defines an interface between BOOST Hydsim and MATLAB® API m-function.	
	MATLAB® DLL * Defines an interface between BOOST Hydsim and MATLAB® DLL (dynamically linked library).	
	PID Controller * Defines Proportional, Integral and Derivative parameters in PID Controller.	
External	**Fire Link (Nozzle) *** Defines an interface between BOOST Hydsim and FIRE nozzle flow cosimulation.	
	Boost Link (Plenum) * Defines an interface between BOOST Hydsim and Plenum element in BOOST.	
	Boost Link (Valve) * Defines an interface between BOOST Hydsim and Valve element in BOOST.	

3.4. Connections

The system model in BOOST Hydsim Workspace is constructed by joining icons with connections. These are represented on the screen as a straight line or a combination of straight lines. Each connection must always join two elements (icons). There are four types of connections available on the **Palette** menu located above the **Element** group menu:

- Hydraulic connection
- Mechanical connection
- Special connection
- Wire connection

3.4.1. Hydraulic Connection

A hydraulic connection is represented as a blue line with an arrow at the end. The arrow points to the positive direction of the fluid flow. This connection can only be used with icons having blue (hydraulic) or black (general-purpose) attachment points. Though hydraulic connections can be arbitrarily oriented on the screen, they always imply one-dimensional flow (along the x-axis). There is no input data associated with a hydraulic connection.

Depending on the type of connected elements, the hydraulic connection may be reversible or irreversible. This is not controlled by the GUI and is the user's responsibility. For an irreversible connection, definition of wrong direction will lead to false interpretation of the model. Refer to the BOOST Hydsim Users Guide for more information.

☞ **Note:** A hydraulic connection is only a symbol, not a hydraulic line (i.e. it cannot represent a pipe or tube).

Normally a hydraulic connection must have one output and one input end. Connections with two output ends or two input ends are formally possible but should never be used.

The direction of connection can be changed by selecting an attachment point (highlight desired arrowhead by clicking left mouse button) and pressing one of the buttons in out (input or output) in the **Palette** menu.

3.4.2. Mechanical Connection

Mechanical connection is represented as a red line with an arrow at the end. The arrow indicates the positive direction of initial motion. This connection can only be used with icons having red (mechanical) or black (general-purpose) attachment points. Mechanical connections are defined in two translational directions (along the x and y-axes) and one rotational direction (around the w axis). As mechanical connections can be arbitrarily oriented on the screen, their actual direction is specified in the input dialog box by clicking an appropriate toggle button x, y or w. Default direction is x.

☞ **Note:** Mechanical connection has its own Input Dialog, which must be supplied with data (preload, stiffness, damping and area). These may be either constant or variable (nonlinear).

Normally a mechanical connection has one output and one input end. Connections having two output ends or two input ends are also possible but should not be used except in special cases.

Depending on the type of connected elements, the mechanical connection may be reversible or irreversible. This is not controlled by the GUI and is the user's responsibility. For an irreversible connection, definition of wrong direction of connection will lead to false results. Refer to the BOOST Hydsim Users Guide for more information. The direction of connection can be changed by selecting an attachment point (highlight desired arrowhead by clicking left mouse button) and pressing one of the buttons in out (input or output) in the **Palette** menu.

3.4.3. Special Connection

 A special connection is represented as a green line. Contrary to the hydraulic and mechanical connections, it has no arrowhead (direction). This connection is exclusively used with icons having green attachment points (semicircles). Special connection establishes a physical or functional dependence of one icon on another. Usually it implies that both icons are parts of a certain physical unit (e.g. plunger and spill port, injection needle and nozzle orifice etc.).

3.4.4. Wire Connection

 A wire connection is represented as an orange line. As special connection, it has no arrowhead (direction). This connection is used with icons having orange attachment points (semicircles). Wire connection establishes a BOOST Hydsim – MATLAB® Interface, i.e. by wire connection, MATLAB® element receives parameters from connected elements and sends its results to them.

3.5. Unit System

AVL-Workspace converts input values from **input units** (user-defined units in Workspace) into **simulation units** (units used by the simulation kernels). In general, **default units** will be used for input values, but you can override the default units for each individual input value in any dialog. Default units can be defined in the **Options | Units** dialog:

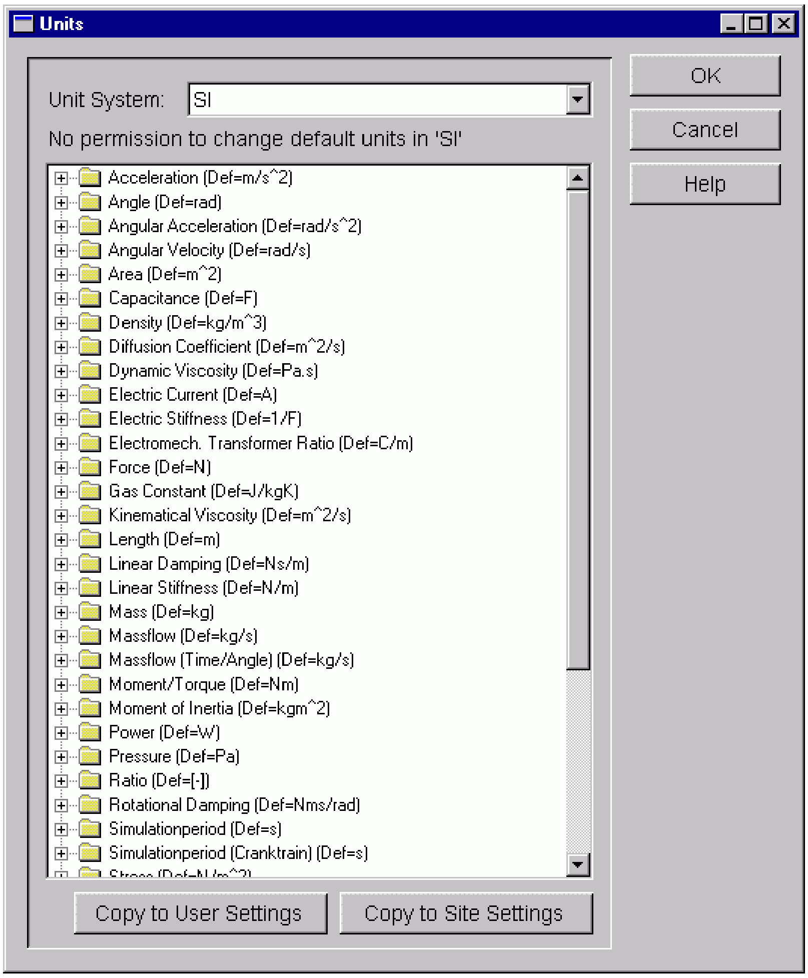


Figure 3-3: Default Unit Dialog

Default unit settings can be adopted from user, site or system level. Settings can be defined at user and site level, either by copying from higher configuration levels (e.g. from site to user level) or by changing individual default input units. The three levels have the following meaning:

- **SI**: settings on default SI units
- **English Metric**: settings on default English metric units
- **N-mm-s**: settings on default N-mm-s units
- **Engine Cycle Simulation**:
- **Site**: settings shared by all users of your site
- **User**: settings for individual users

To change default input units, expand the relevant unit type node and double click the desired unit. A gray marker will appear next to the new default input unit.

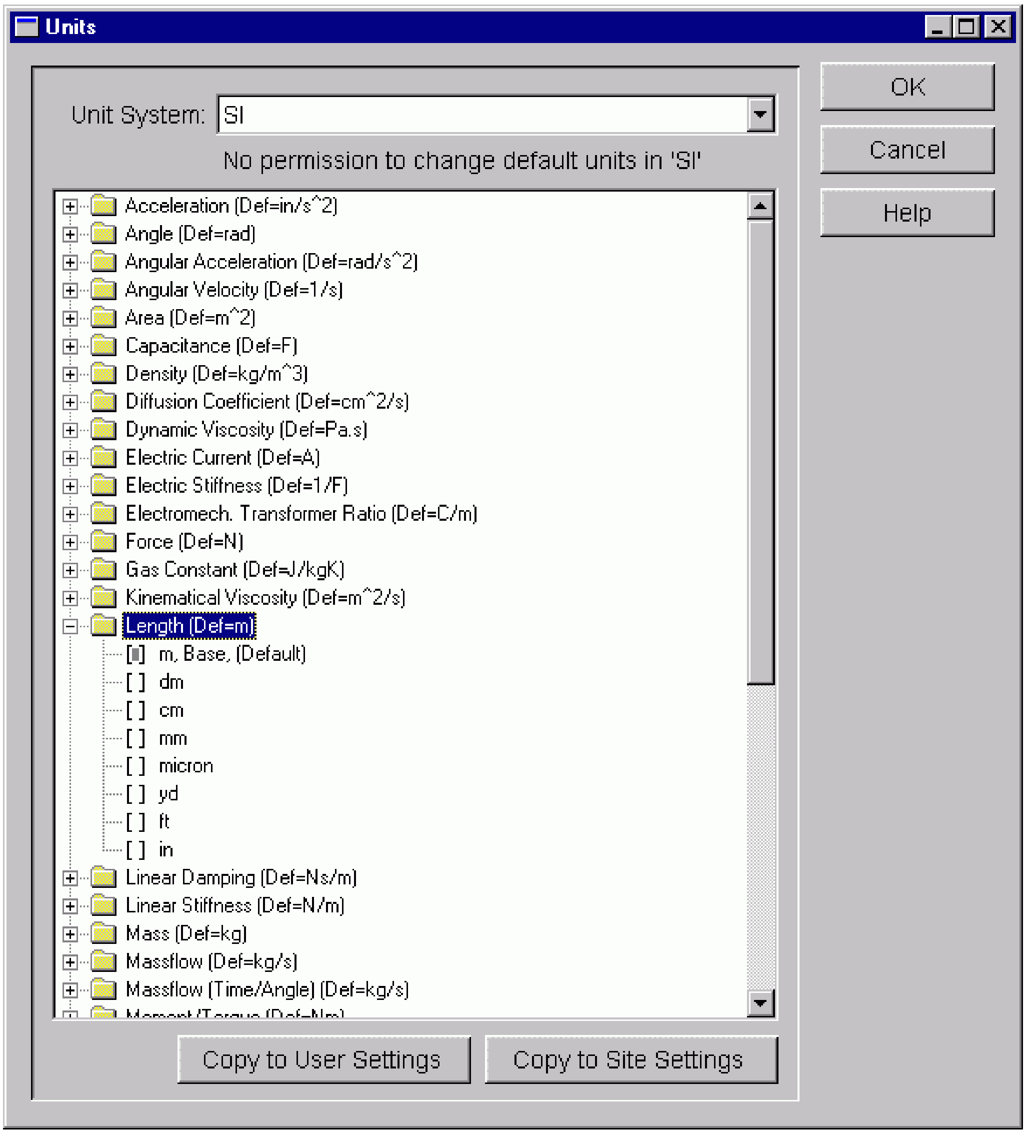


Figure 3-4: Unit Tree

3.6. Run Simulation

Select **Simulation | Run** to start a simulation run (*Figure 3-5*) or click the appropriate icon (*Table 3-2*).

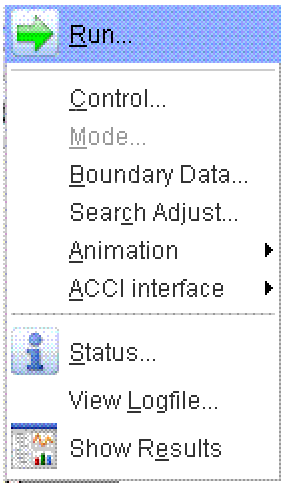


Figure 3-5: Simulation Menu

Table 3-2: Running icons

ICON	OPTION	DESCRIPTION
	Run Simulation	Run (or Restart) active Case
	Simulation Status	Open dialog for Simulation Status

3.6.1. Run Simulation

Selecting **Run** from the **Simulation** pull-down menu opens the Run dialog as shown in *Figure 3-6*. Select appropriate cases and click **Run** to start or restart calculation. A simulation status window pops up giving an overview of the status of the simulation (*Figure 3-8*).

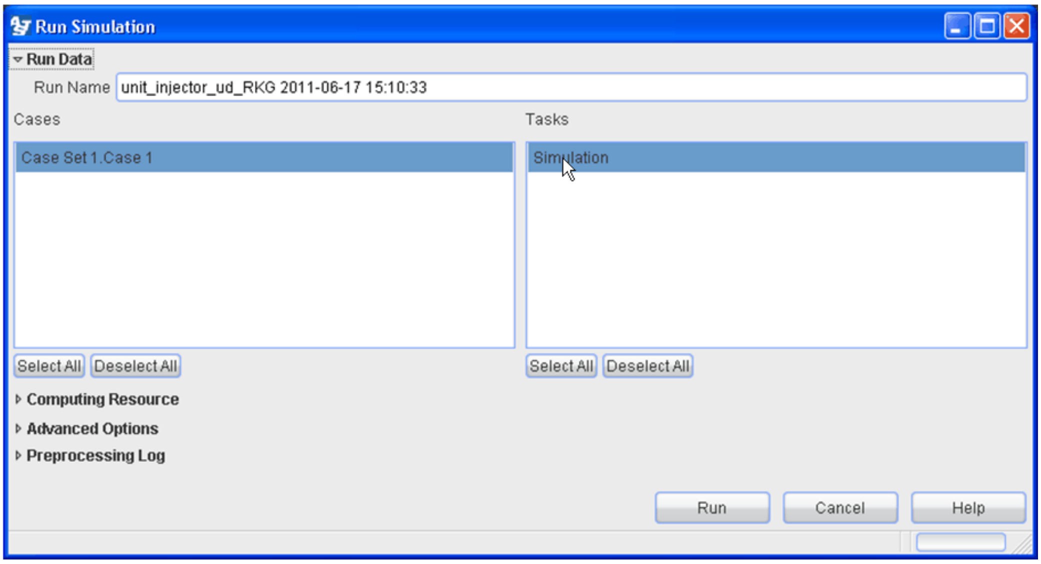


Figure 3-6: Run Simulation Dialog Box

Click  to start the BOOST Hydsim kernel as a background process for the active Case. Which kernel (**Fixed-step** or **Variable-step**) is started depends on the selected solver option in **Simulation Control** dialog (*Figure 3-7*). A simulation status window pops up giving an overview of the status of the simulation (*Figure 3-8*).

By default **Run** command starts standard simulation. However, if **Restart Calculation** check box is checked in **Simulation Control** dialog (*Figure 3-7*), **Run** command will restart (continue) simulation of the model from the previously saved position within the extended (specified anew) time or angle interval (refer to *Section 6.1.5* for detailed information). **Restart Calculation** check box is currently available only for the **Fixed-step** solver.

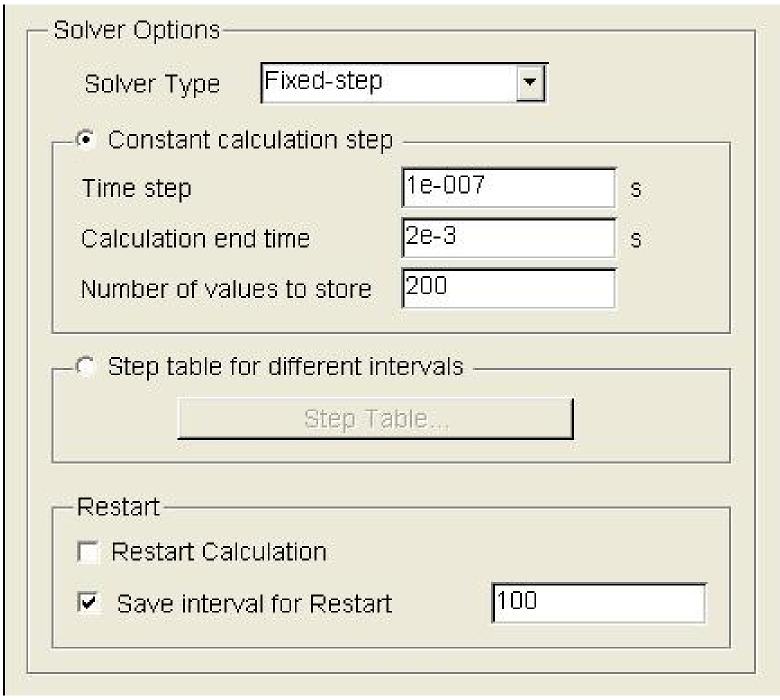


Figure 3-7: Calculation Control Dialog part with Solver Options

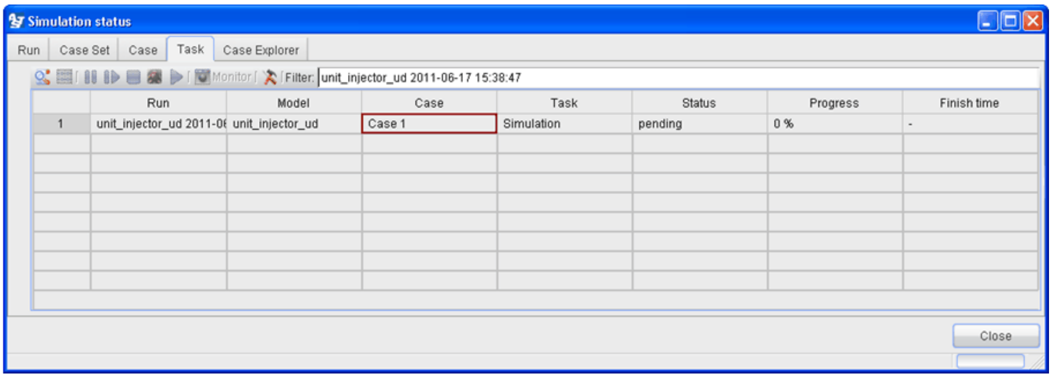


Figure 3-8: Simulation Status

Refer to the Online **Help** for more information on buttons for process control.

3.6.2. View Logfile

More detailed information on the simulation run, as produced by the simulation kernel, can be monitored by selecting **View Logfile** (refer to *Section 4.4* for detailed information).

3.6.3. Show Results

Select **Show Results** from the **Simulation** pull-down menu to directly open the IMPRESS Chart window with the actual model results directory (refer to *Section 6.3.2* for detailed information).

3.7. Variation of Parameters

This chapter describes how to benefit from varying parameters in BOOST Hydsim models. In most AVL Workspace dialogs it is possible to assign parameters to input fields. That is, constant values of input fields can be replaced by parameter values defined in some global place. Possible applications of global parameters include:

- Variation of parameters, as will be explained later in the description of **Case Explorer**
- Definition of commonly used values in one place

Aside from global parameters, element-specific (local) parameters can be introduced, extending the applicability of parameters. Local parameters are used for:

- Creating simplified and protected model views
- Overriding commonly defined values by element-specific, local values.

Refer to *Section 4.7* for detailed information.

3.8. Case Explorer

With the Case Explorer parameter variations can be easily defined. To start the Case Explorer, select **Case Explorer** from menu **Model** (refer to *Section 4.8* for detailed information).

3.9. Sub-directory for BOOST Hydsim Files

It is recommended to create a product-specific subdirectory …`/boost_hyd and` store all BOOST Hydsim files <model name>.hyd there. This will enable more convenient representation of the post-processor Impress Chart element tree and use of advanced Impress Chart features. Alternatively an old subdirectory …`/hydsim` can be used.

3.10. Online Help

Online help is available for LINUX workstations through Netscape Browser for all BOOST Hydsim dialogs. On Windows platforms (XP, Windows 7 etc.) the online help is accessible through Netscape and Internet Explorer (depending on the user's choice).

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:

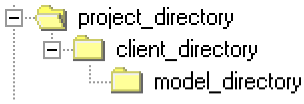


Figure 4-1: Project Structure

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.

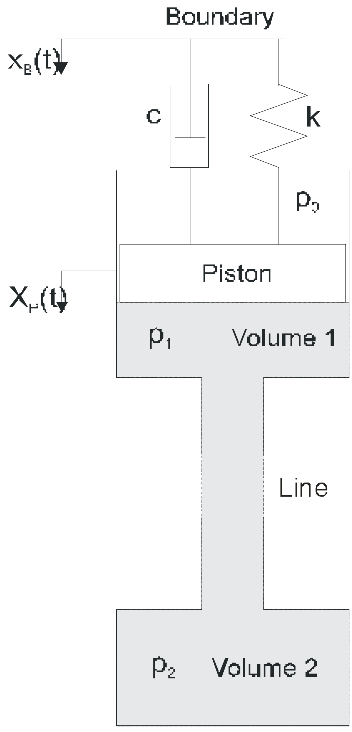


Figure 4-2: Schematic of Simple Hydraulic System

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.

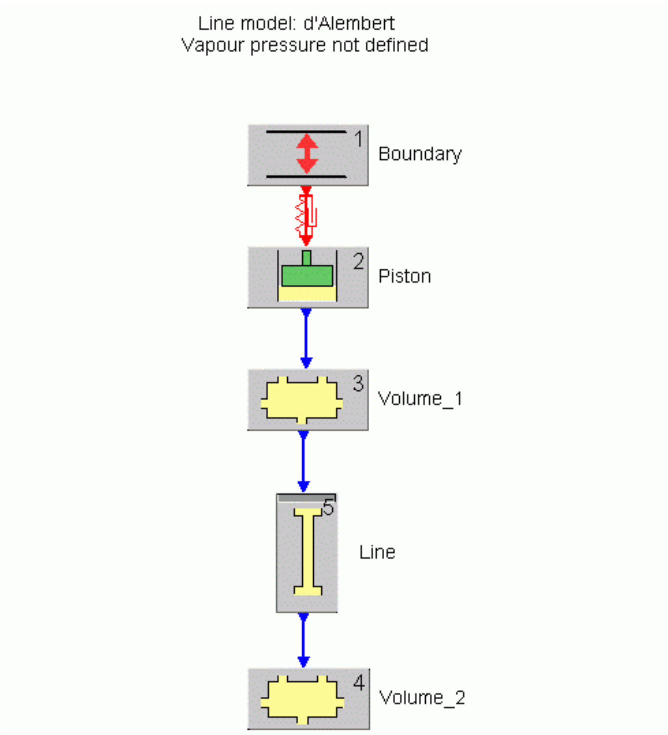


Figure 4-3: BOOST Hydsim Model of a Simple Hydraulic System

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:

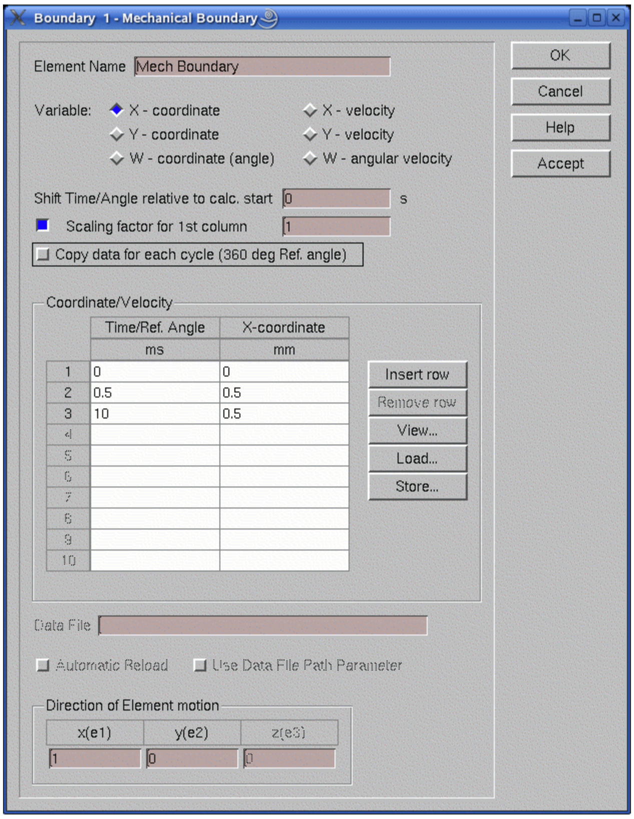


Figure 4-4: Mechanical Boundary Input Dialog

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*.

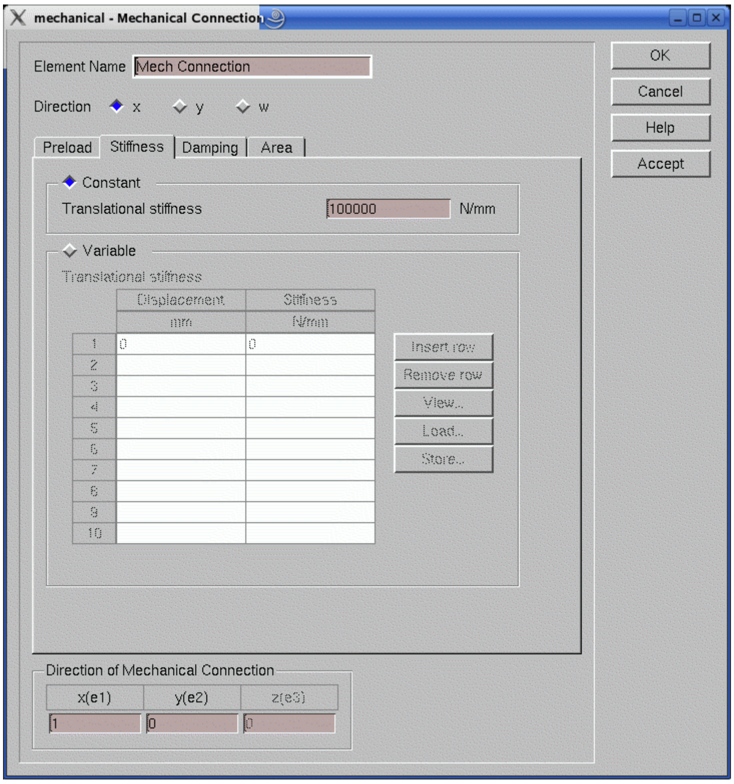


Figure 4-5: Input Dialog of Mechanical Connection

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:

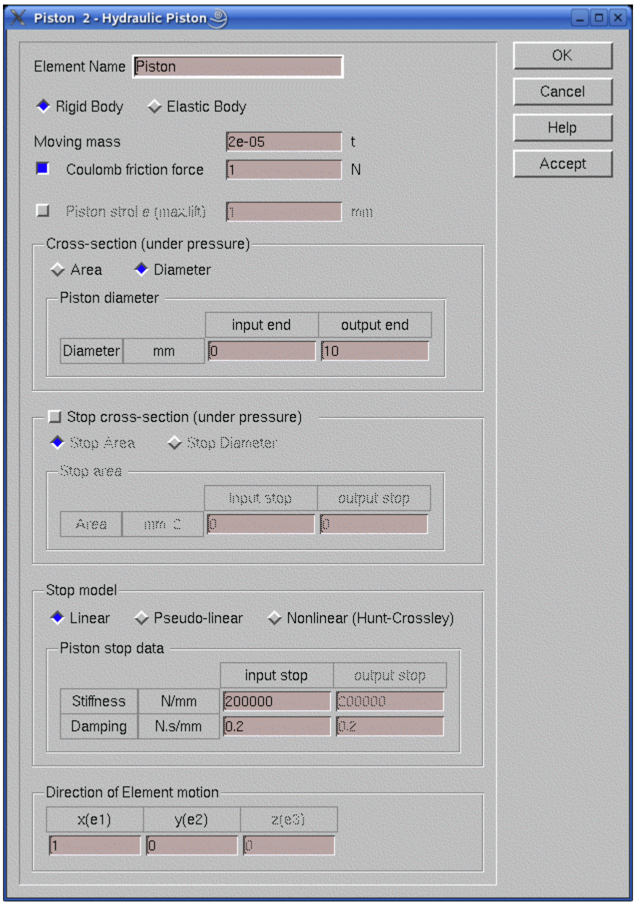


Figure 4-6: Input Dialog of Standard Piston

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:

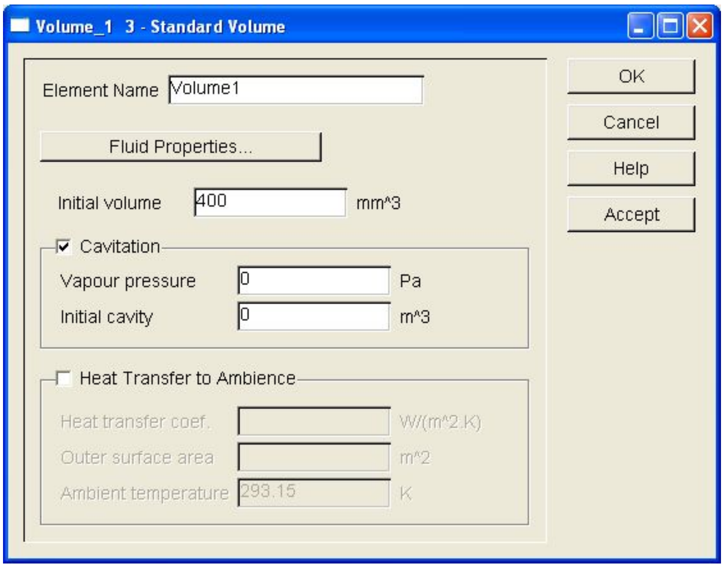


Figure 4-7: Standard Volume Input Dialog

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.

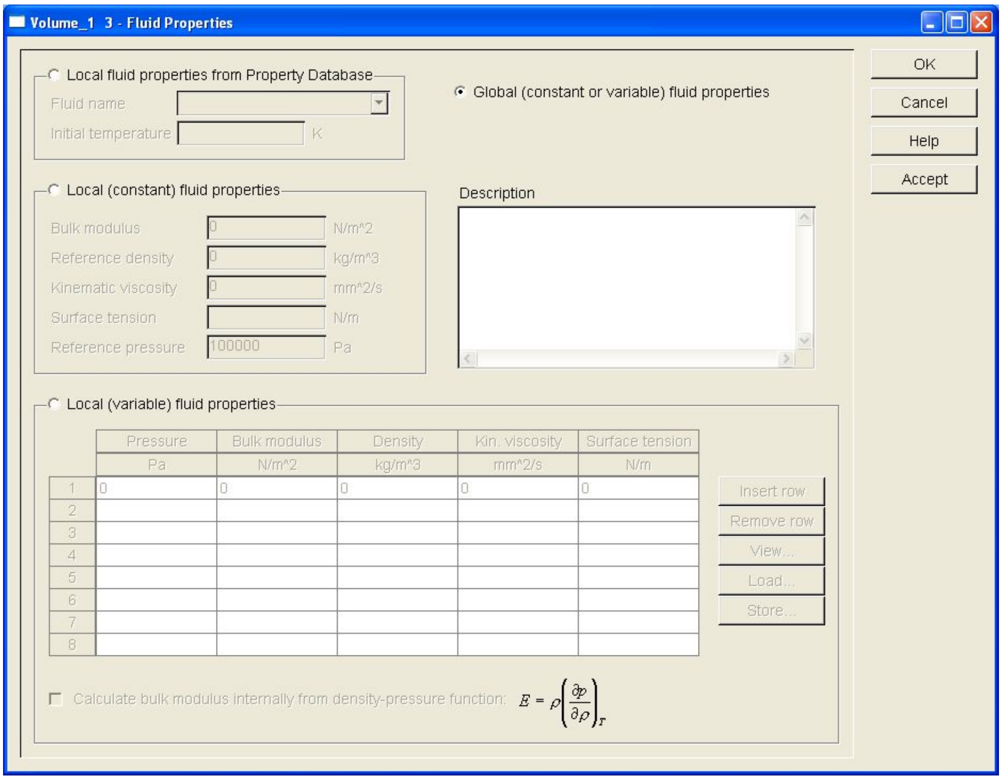


Figure 4-8: Input Dialog of Local Fluid Properties

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:

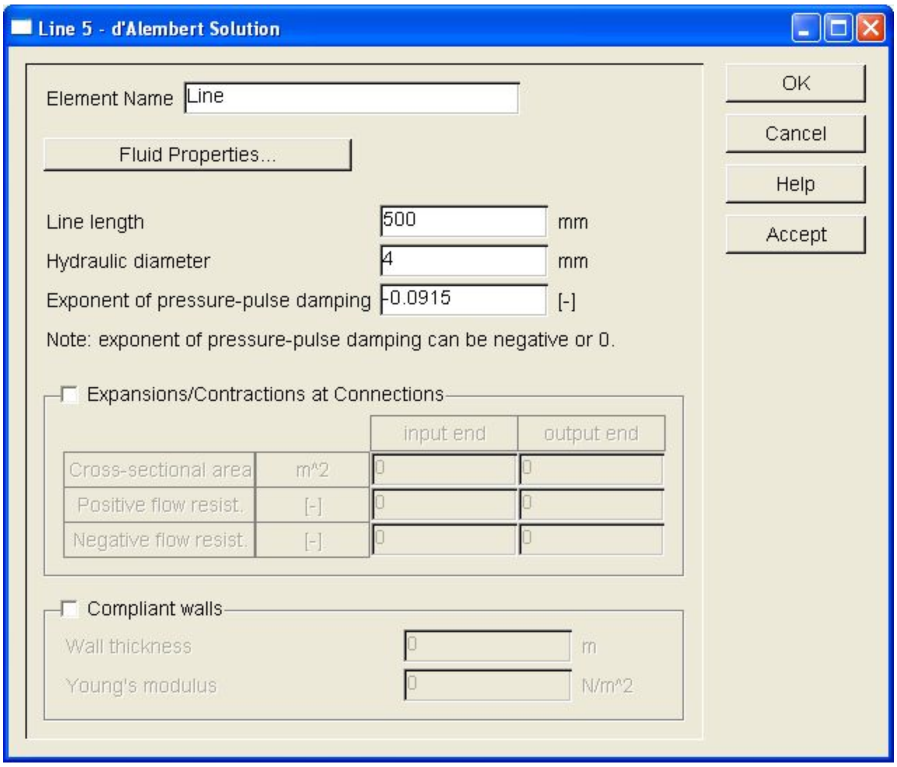


Figure 4-9: Input Dialog of d'Alembert Line

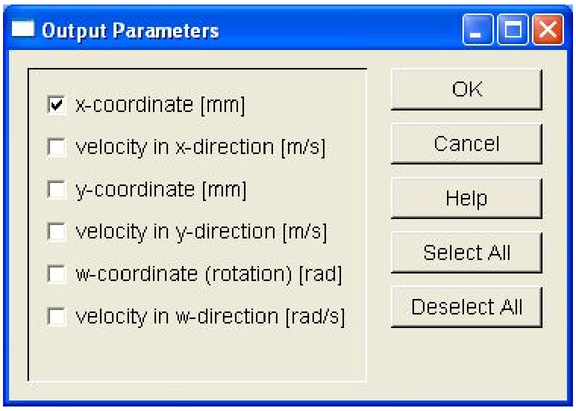
a) The default name of the element is `Line`. The required input parameters are line length, hydraulic diameter and the exponent of pressure-pulse damping. The latter must be negative as it defines the rate of exponential pressure decay along the line (for more detailed information, refer to the BOOST 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.

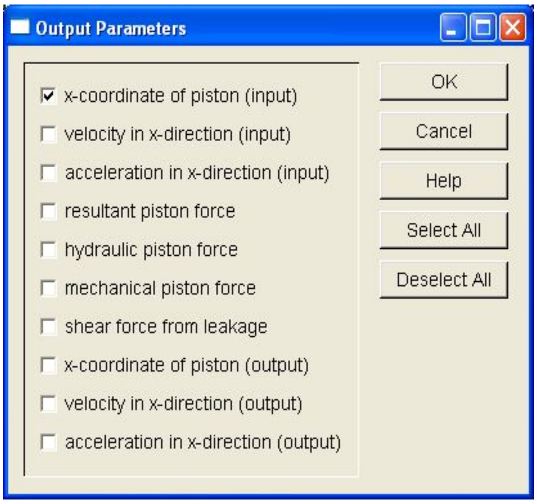
4.3. Output Parameters

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:

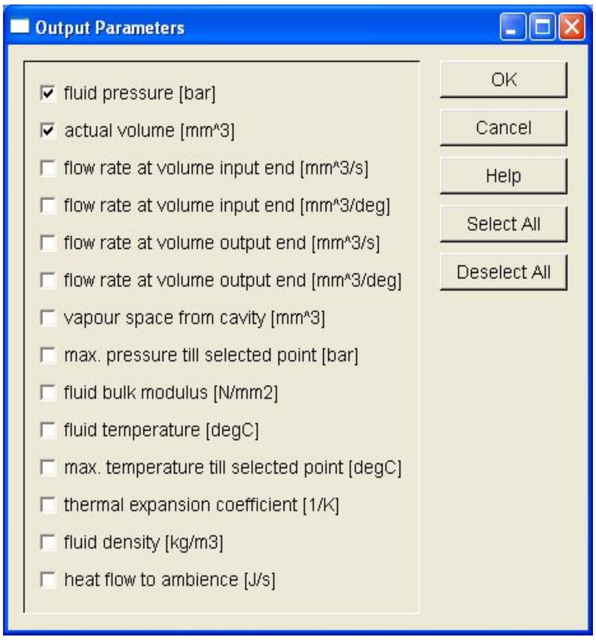
1. Select the element with the mouse.
2. Select **Element | Store Results** to open a window (as shown in *Figure 4-10*) where the appropriate button must be selected so the corresponding variable is stored as output. Selecting **Select All** will store on file all available output parameters listed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

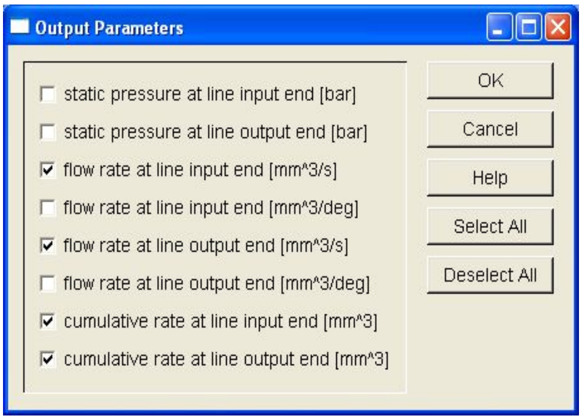


Mechanical Boundary Output Data



Standard Piston Output Data





d'Alembert Line Output Data

Standard Volume 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:

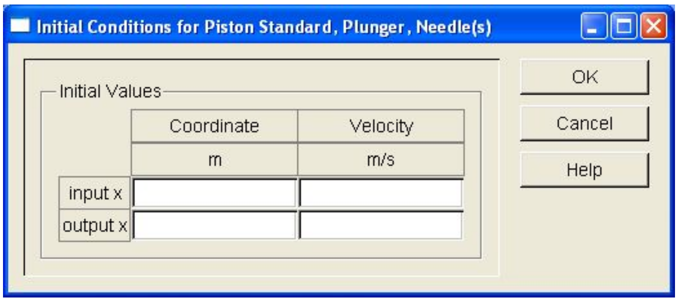


Figure 4-11: Table of Initial Values for Piston, Plunger or Needle

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 .

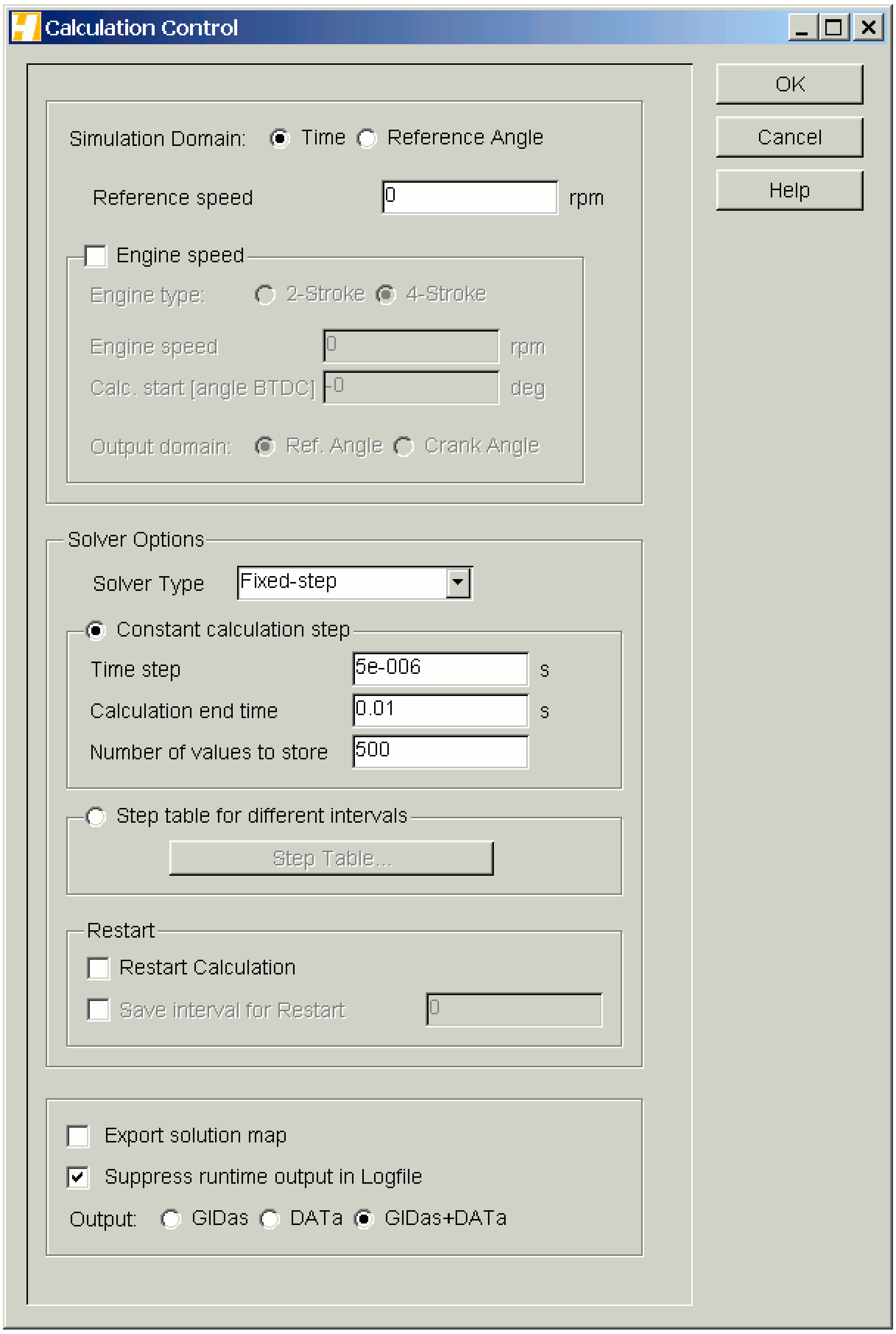


Figure 4-12: Calculation Control Dialog

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} < 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:

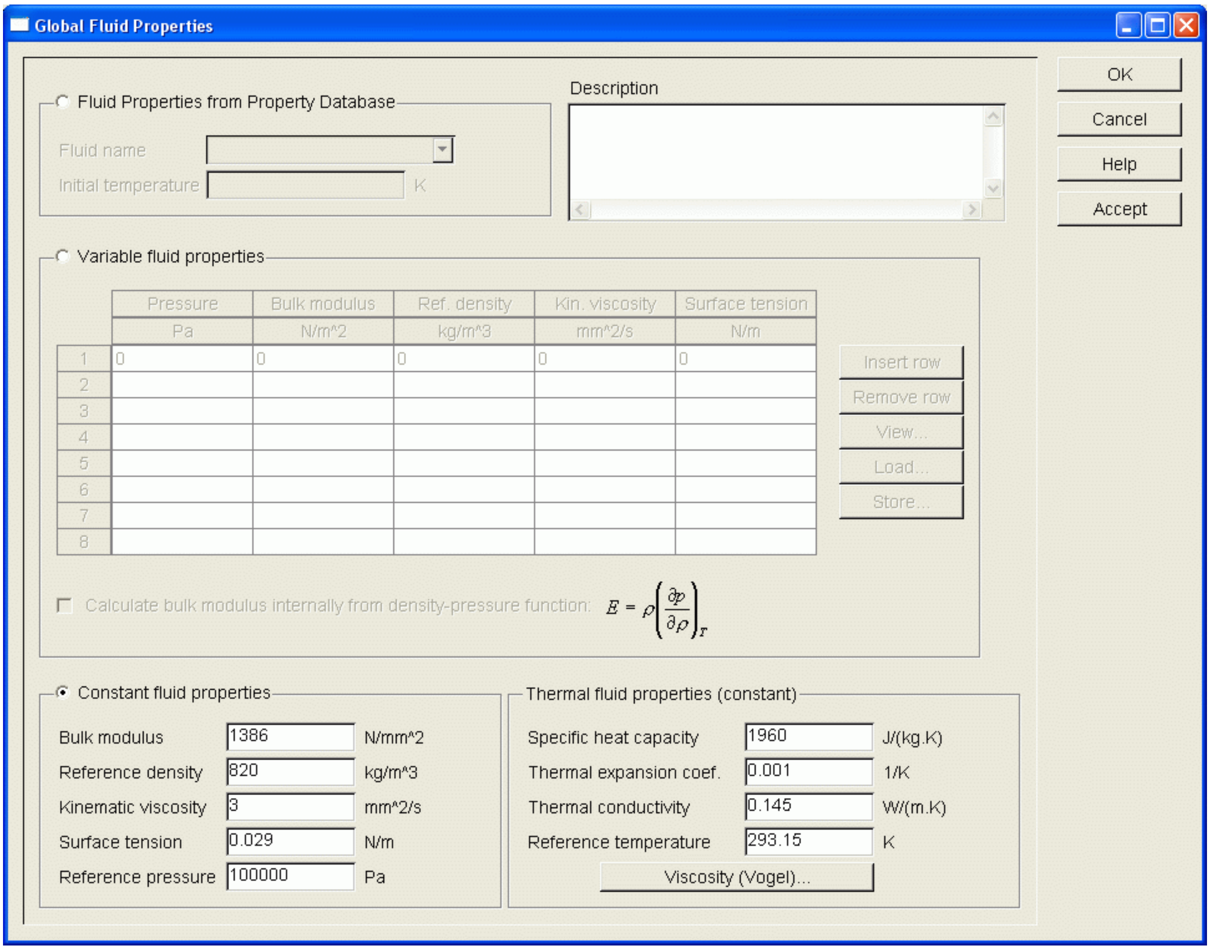


Figure 4-13: Input Dialog of Global Fluid Properties

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*).

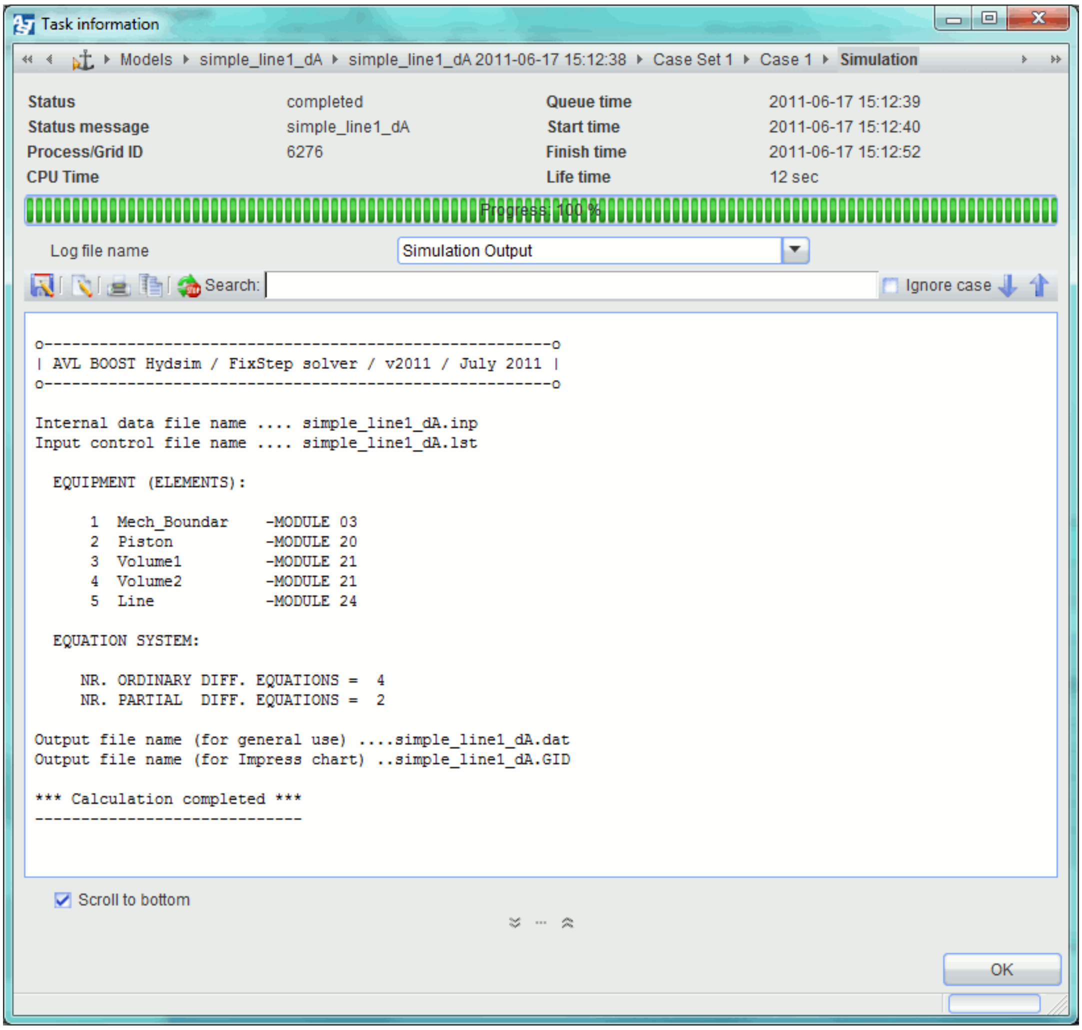


Figure 4-14: Task Information Window

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:

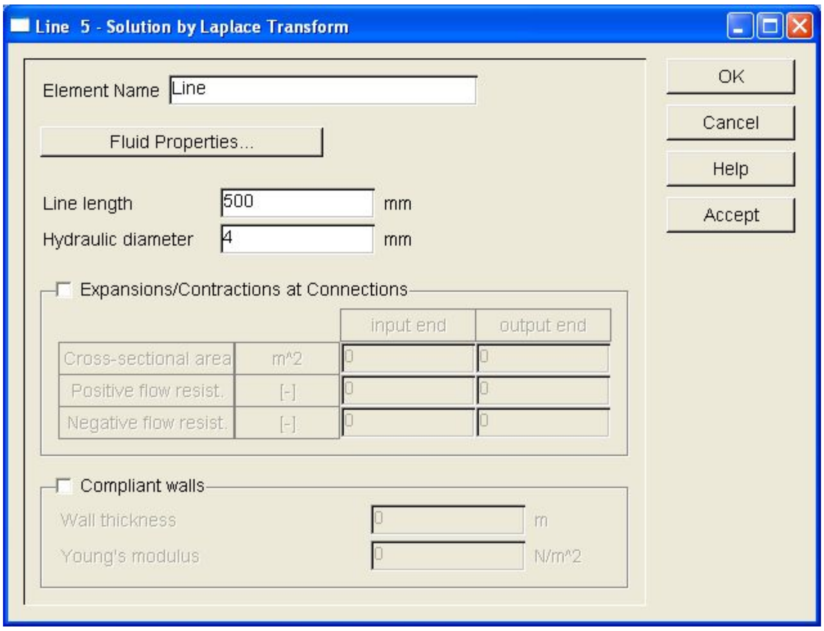


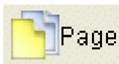
Figure 4-15: Input Dialog of Line (Laplace Transform)

4. 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

1. 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.

2. 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 ( 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*.

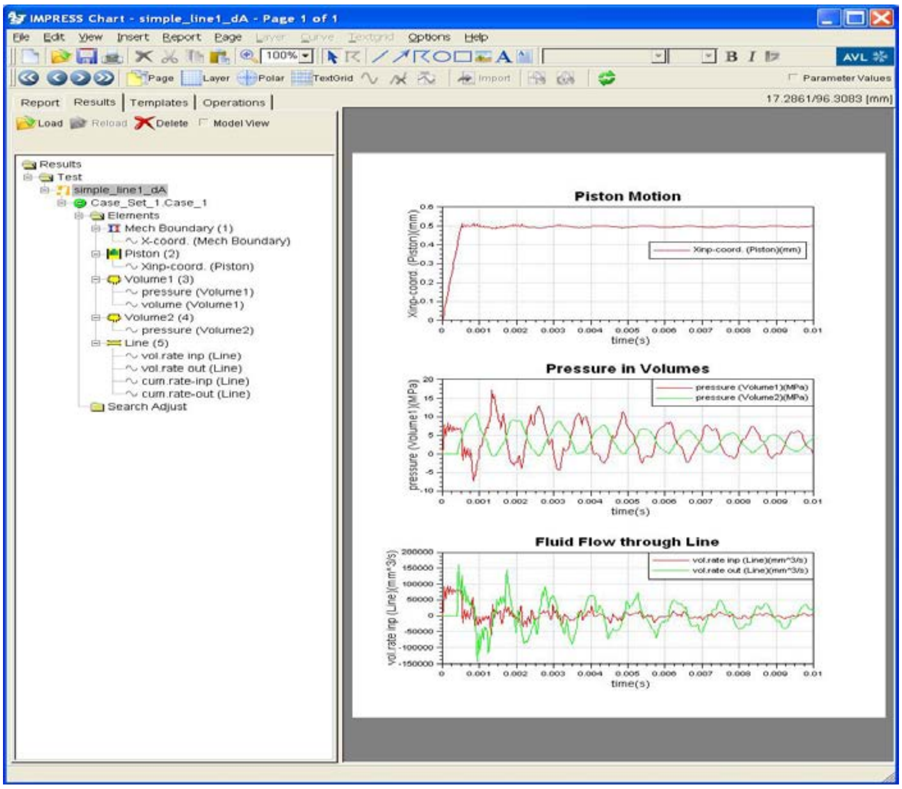


Figure 4-16: Calculation Results without Cavitation Model (zero vapour pressure)

> **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 

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:

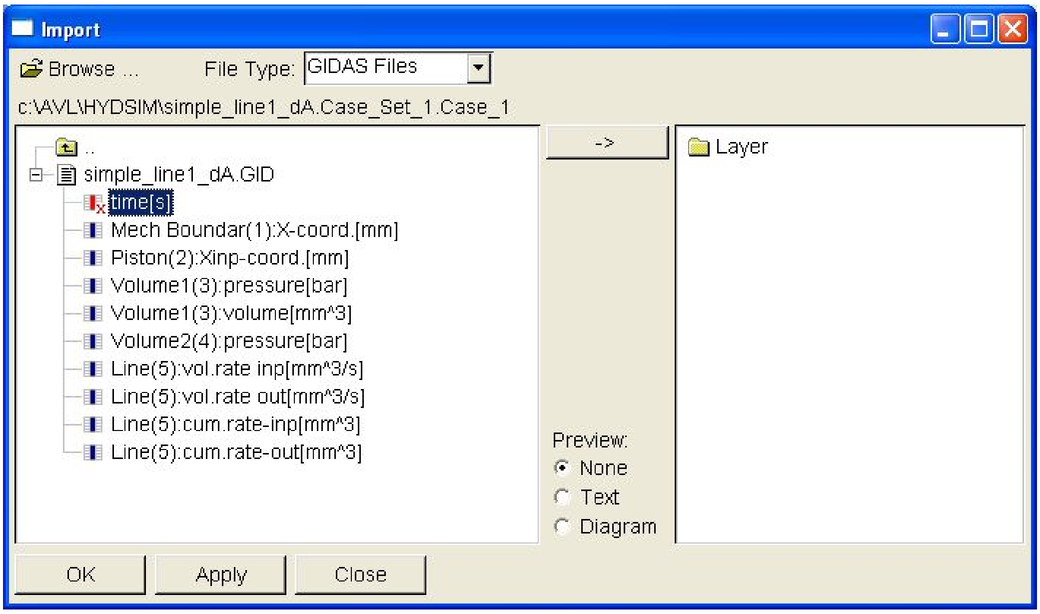


Figure 4-17: Import Window

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:

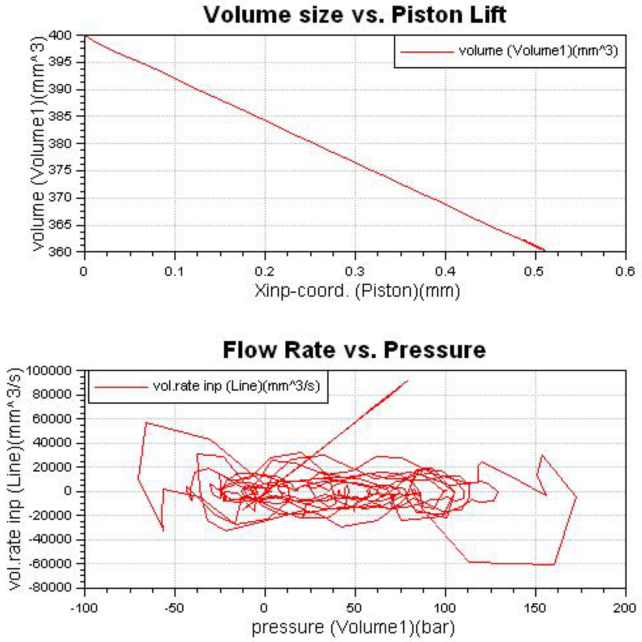


Figure 4-18: Specific Diagrams with Different Variables for X-axis

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.

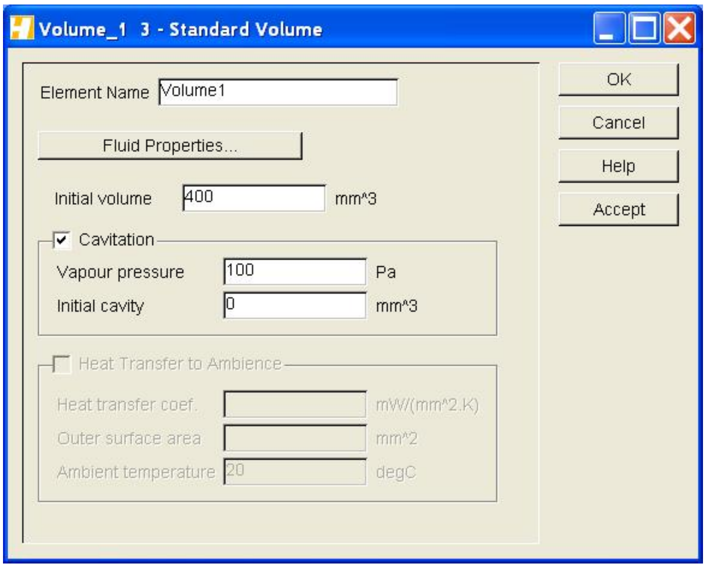


Figure 4-19: Input Dialog of Standard Volume with Vapour Pressure

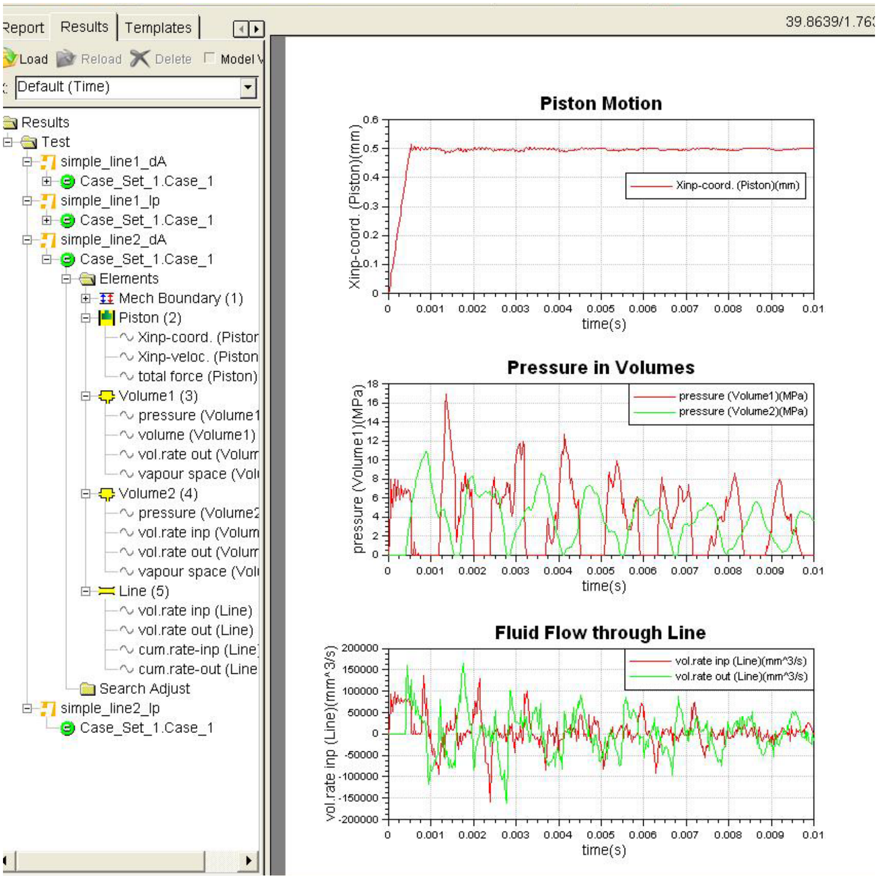


Figure 4-20: Calculation Results with Cavitation Model

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.

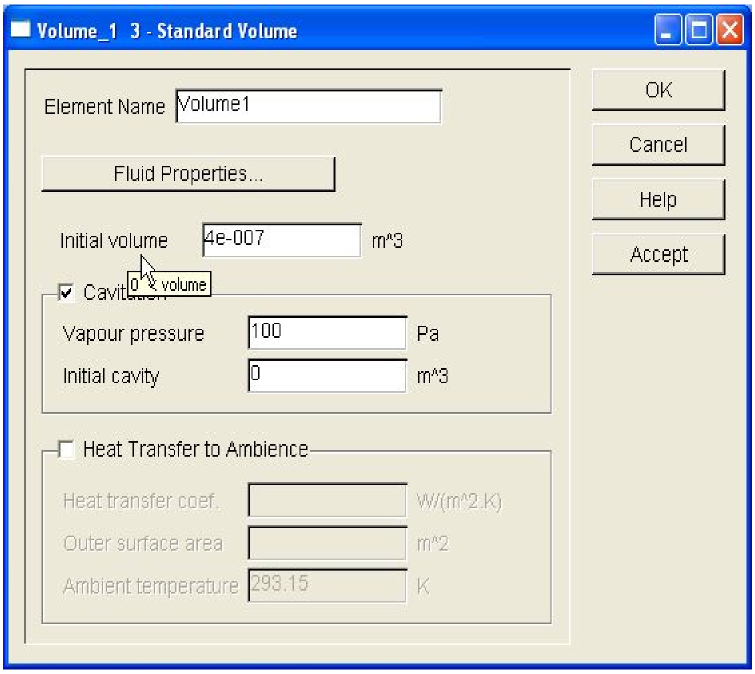


Figure 4-21: Assigning Parameters

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 mm^3 in this case).

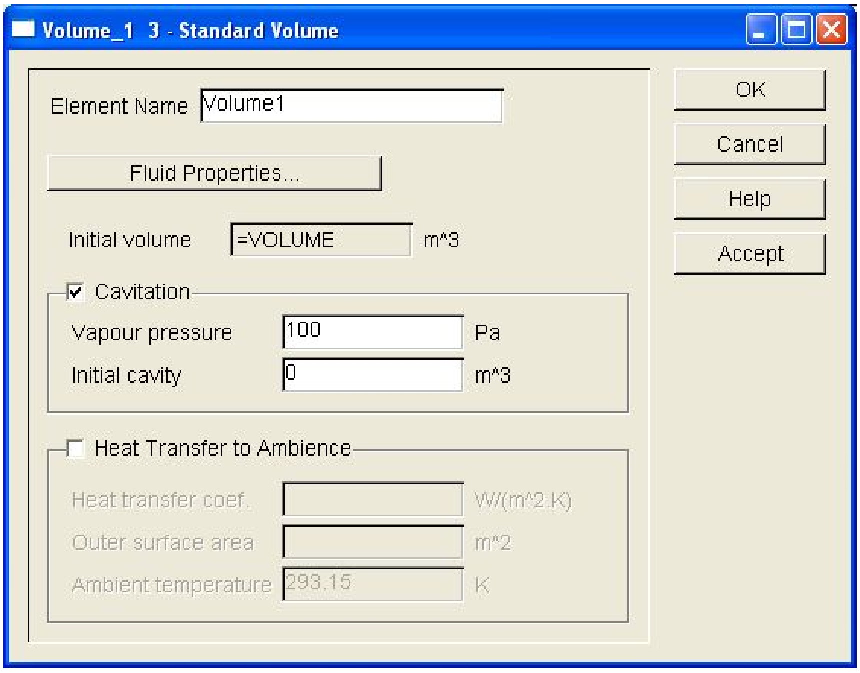


Figure 4-23: Dialog with Assigned Parameter (Variable)

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.

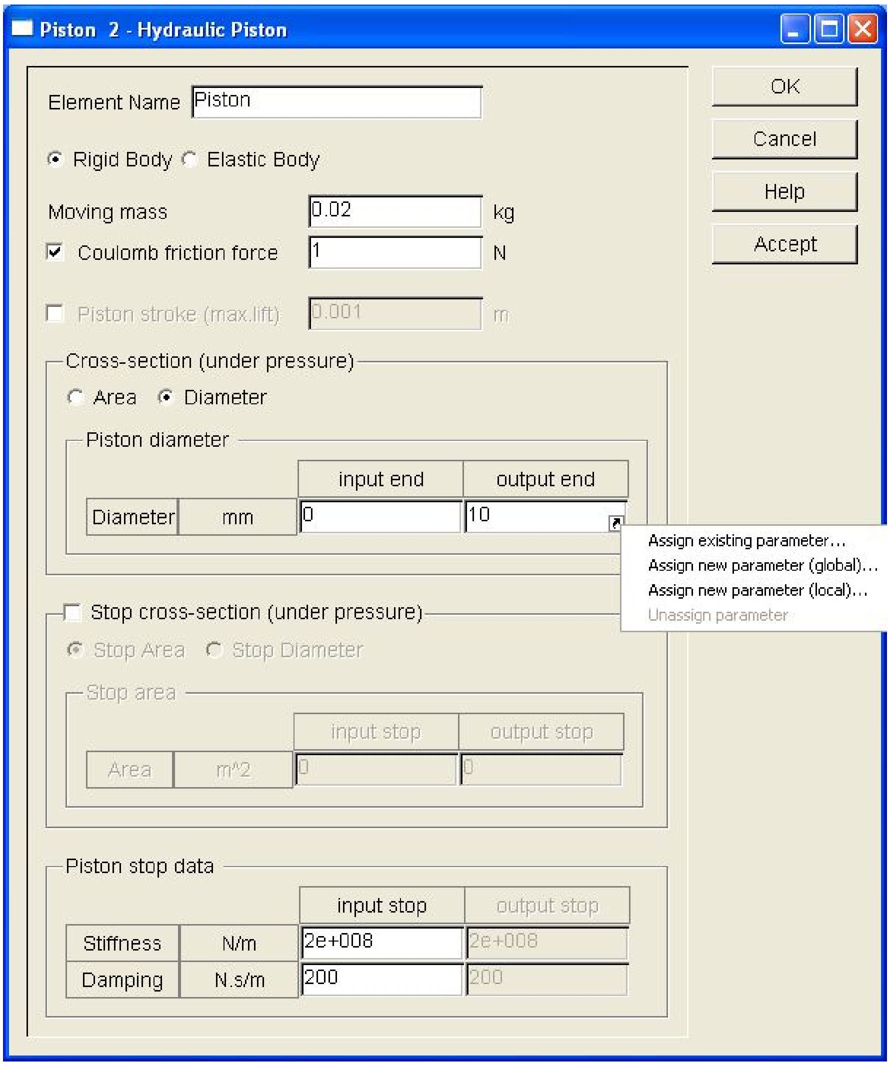
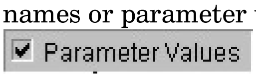


Figure 4-24: Assigning Parameters for a Vector/Matrix Element

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 

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*).

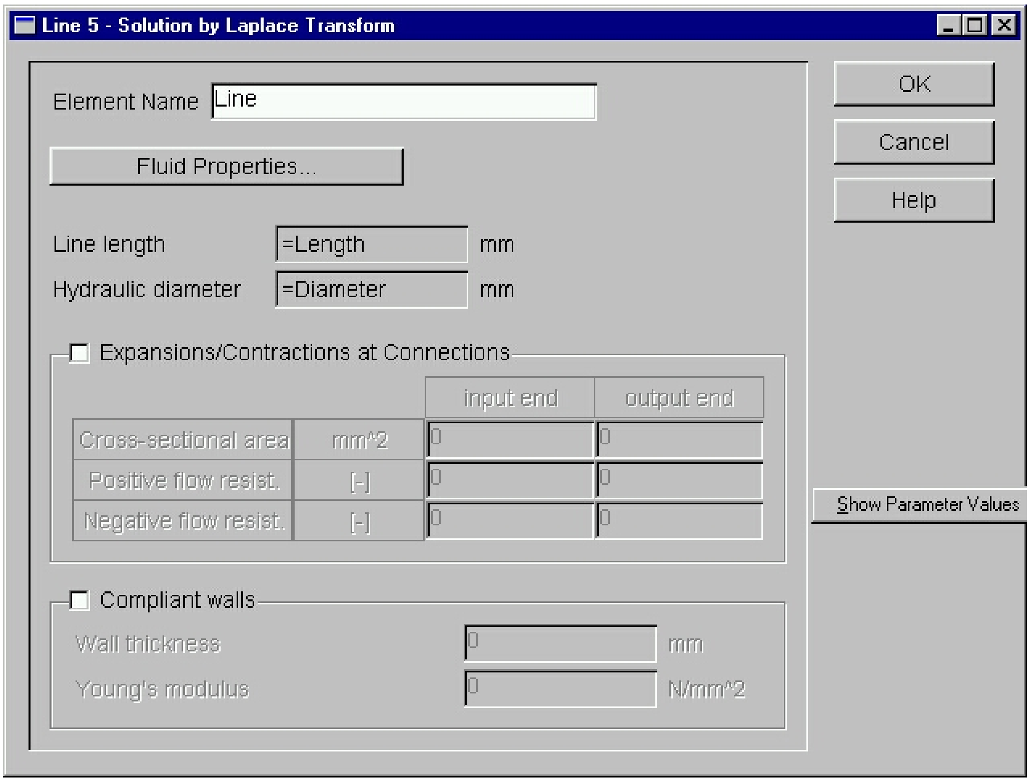


Figure 4-25: Show Values of Assigned Parameters

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:

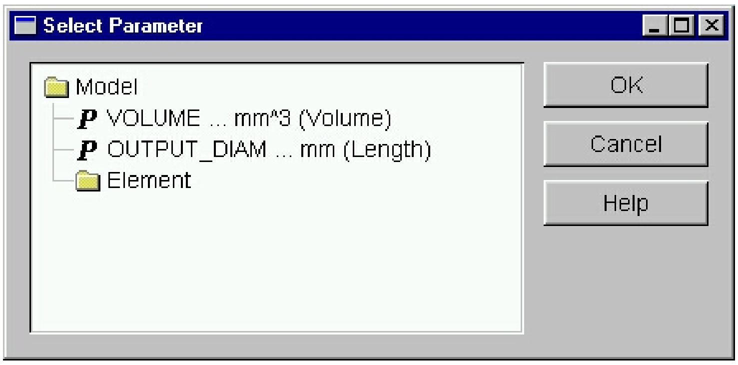


Figure 4-26: Select Parameter Dialog

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:

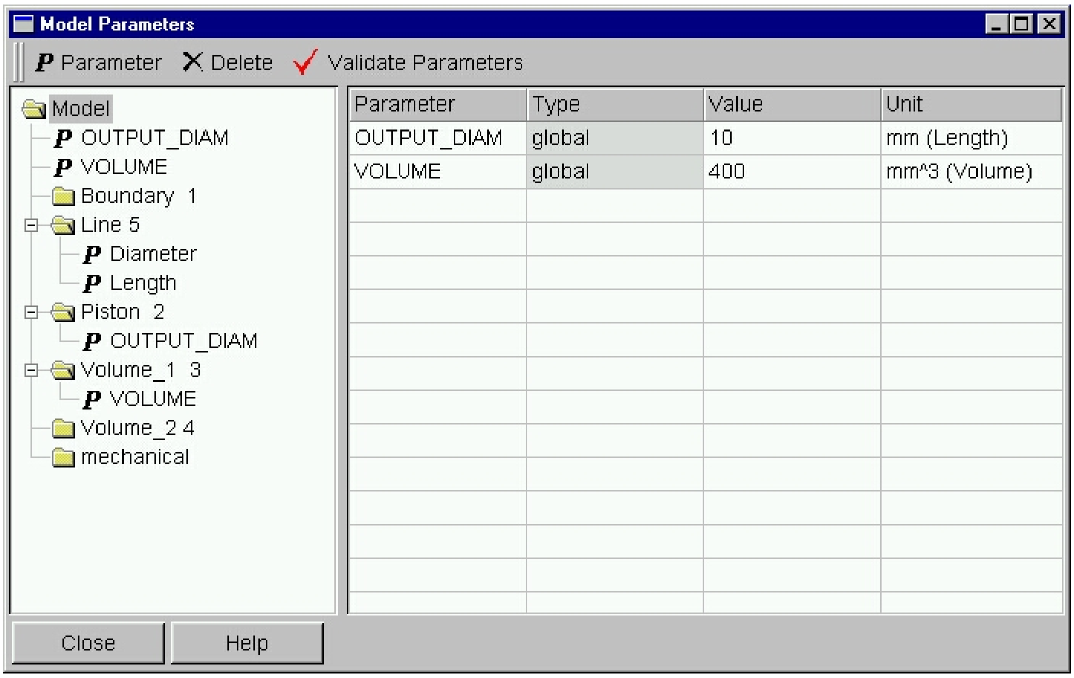


Figure 4-27: Model Parameters Window

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:

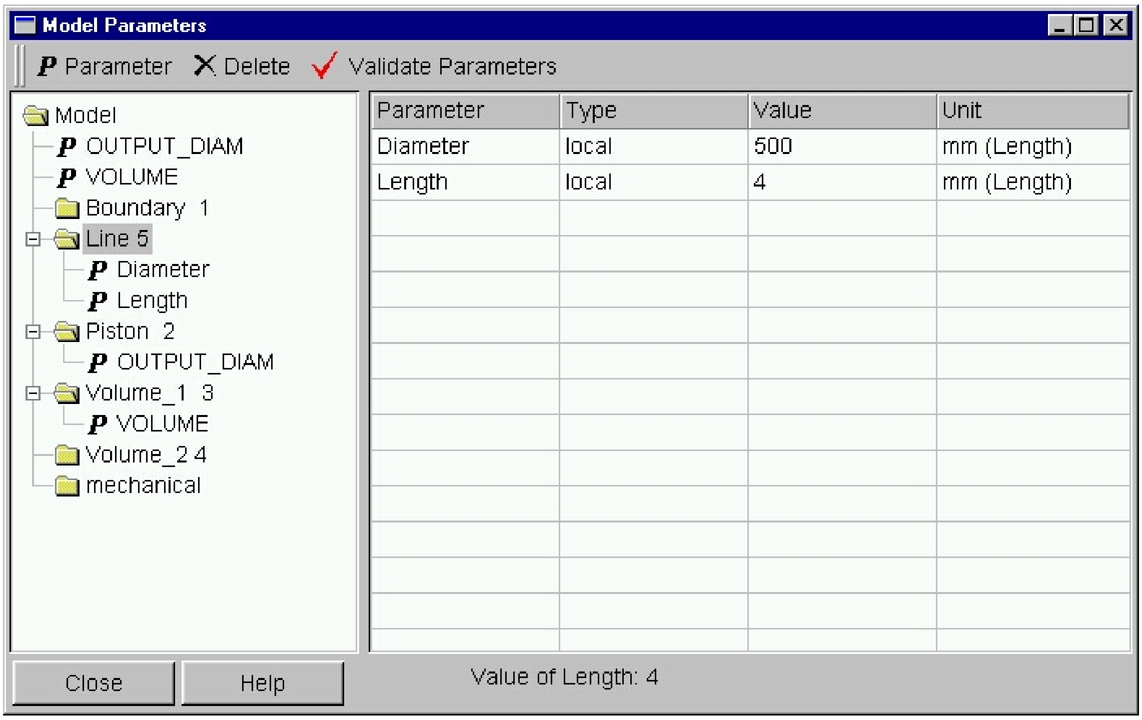


Figure 4-28: Parameters of the Selected Element (Line)

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*).

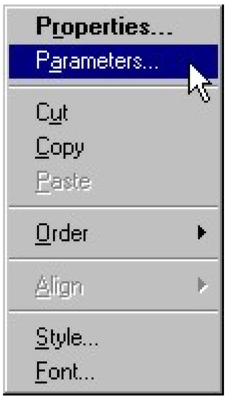


Figure 4-30: Element Pop-up Menu

> ☞ **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 ✕ 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:

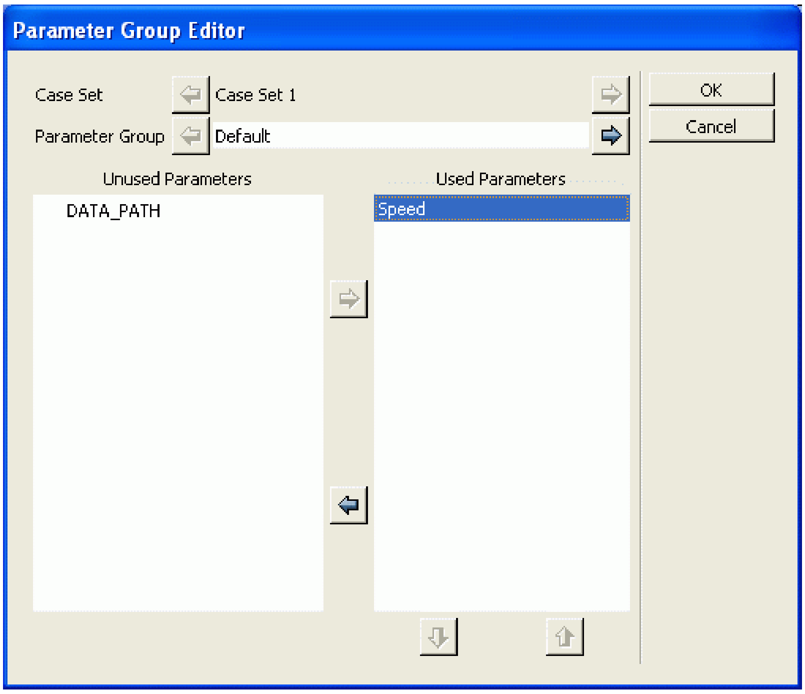


Figure 4-31: Parameter Group Editor Dialog

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.

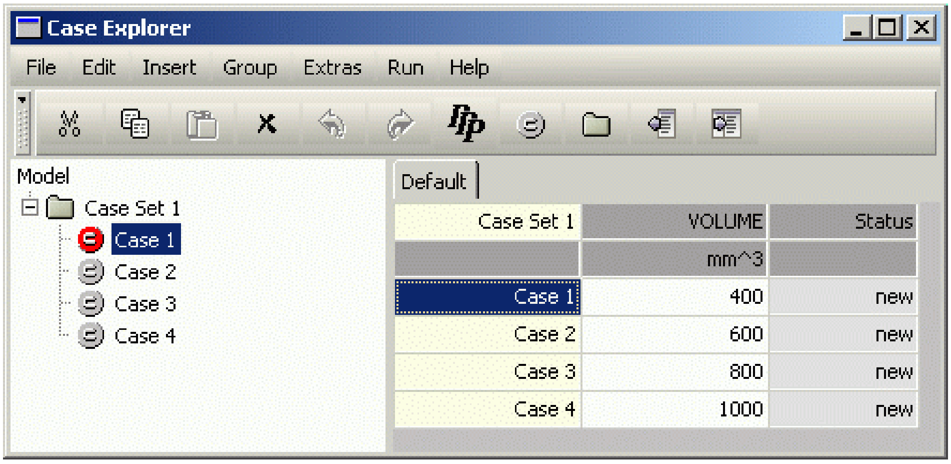


Figure 4-32: Case Explorer Window

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:

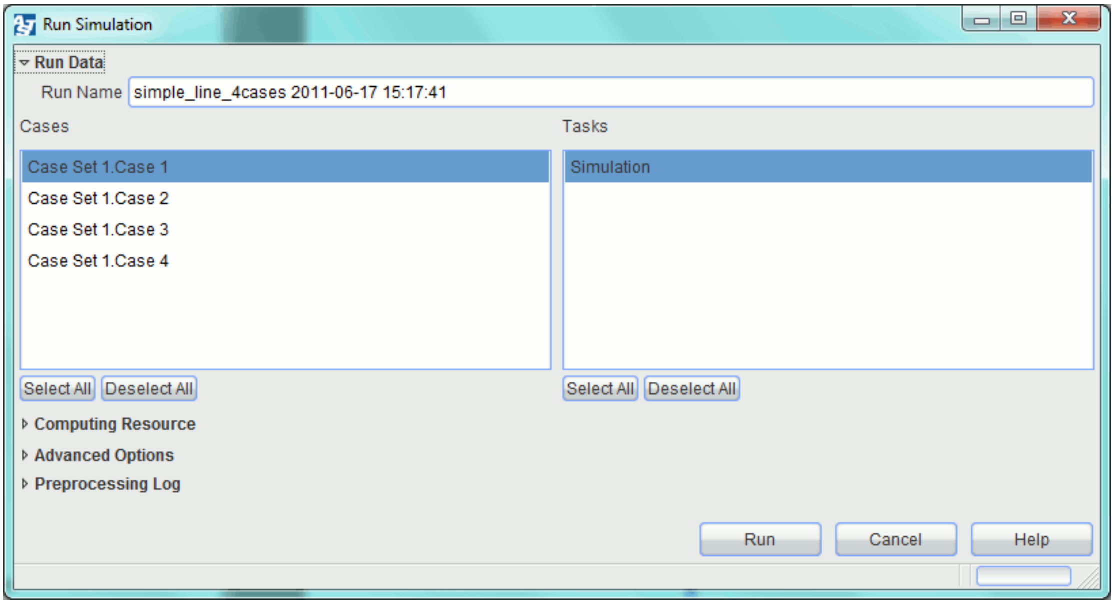


Figure 4-33: Run Dialog with Multiple Cases

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.

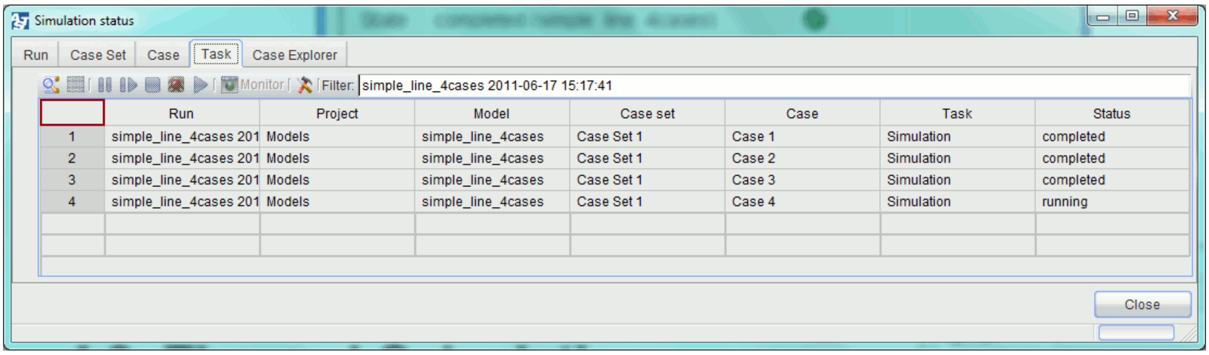


Figure 4-34: Simulation status with running Case 4

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**.

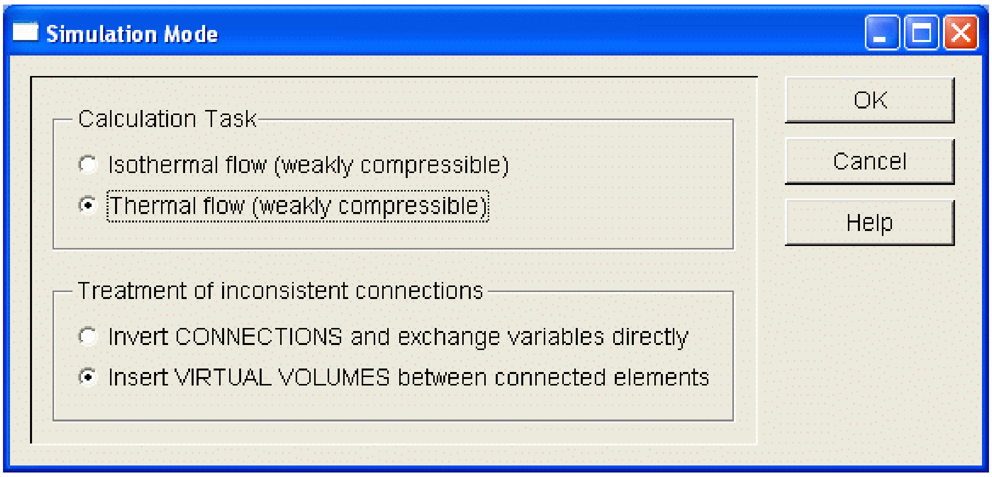


Figure 4-35: Simulation Mode dialog

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.

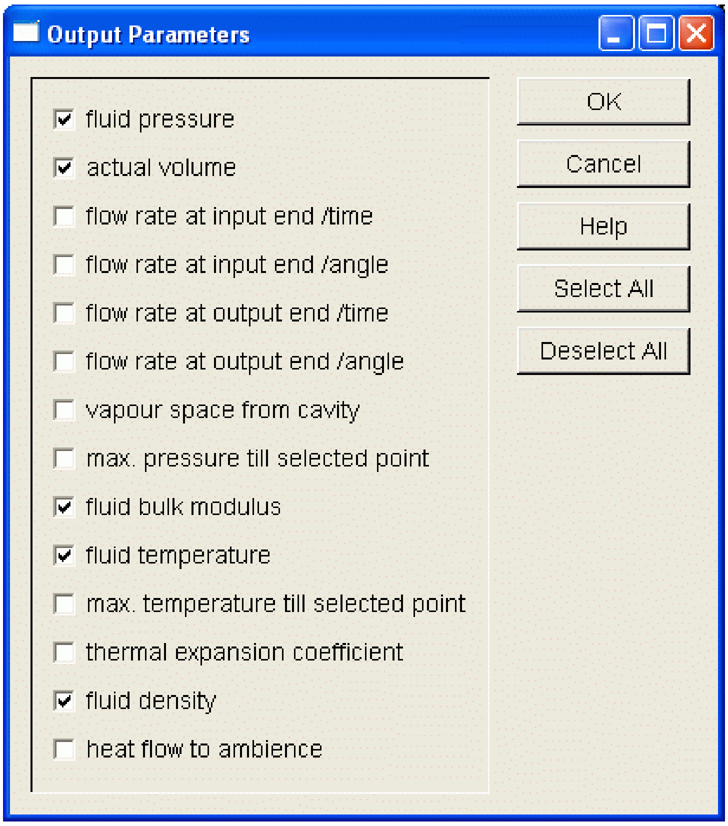


Figure 4-36: Volume output parameters

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.

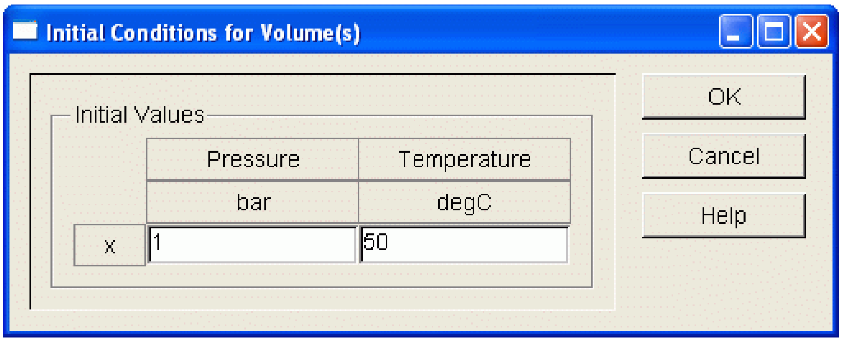


Figure 4-37: Volume initial conditions

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*.

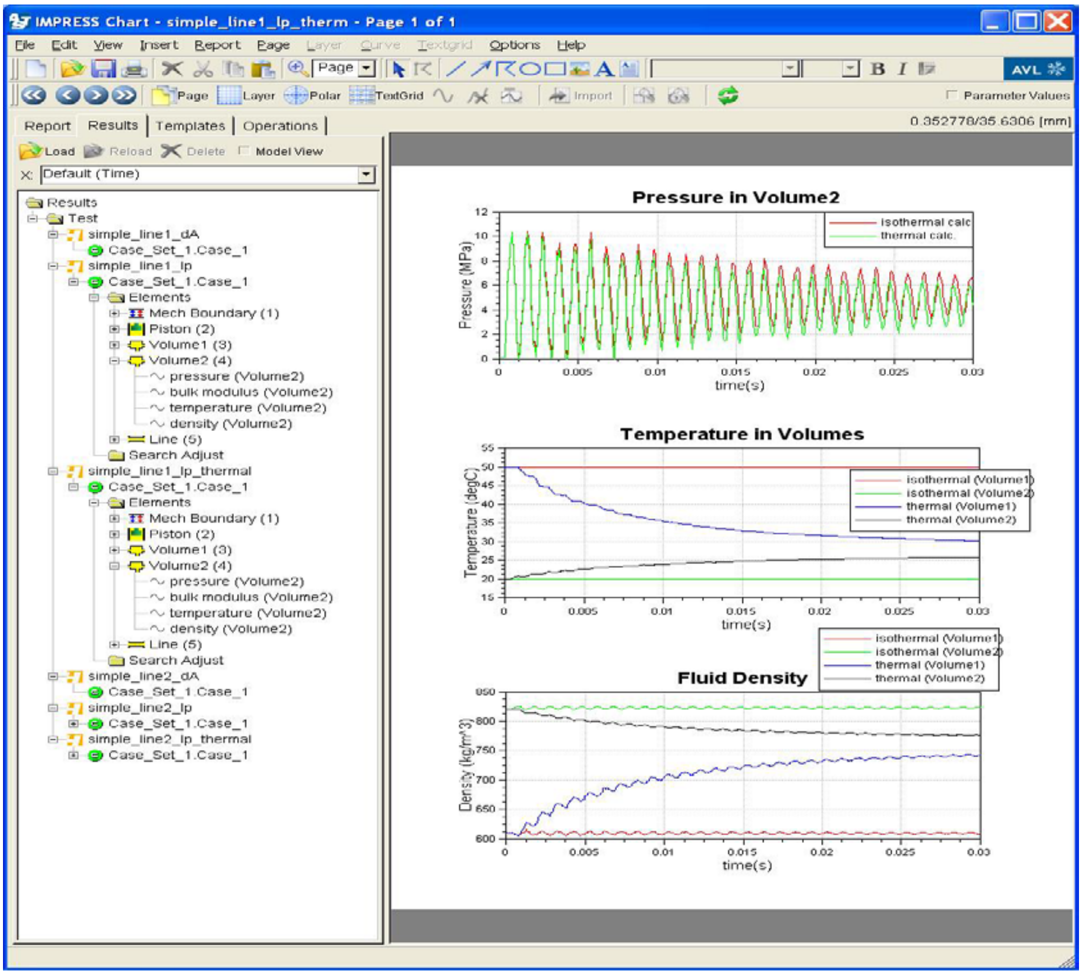


Figure 4-38: Result comparison between isothermal and thermal calculation

5. COMMON RAIL

This chapter is aimed at guiding the user through the modeling of typical common rail injectors controlled by two-way and three-way solenoid valves. These are the models of realistic diesel injection systems.

5.1. Injector with a Two-way Solenoid Valve

In the previous chapter, the basic modeling steps were described which are necessary to create a BOOST Hydsim model. These principles apply for modeling of any hydraulic system, including common rail, and will not be repeated here. However, with the growing complexity of the model, new element types are introduced and additional modeling tools must be employed for building up the model of a real system. Furthermore, special care must be taken in branching the system, specifying element connections with each other and to the surrounding environment, etc. This will be demonstrated by creating a model of a common rail injector fitted with a two-way solenoid control valve.

5.1.1. Overview

A typical common rail injector with 2/2-way solenoid valve is shown in *Figure 5-1*.

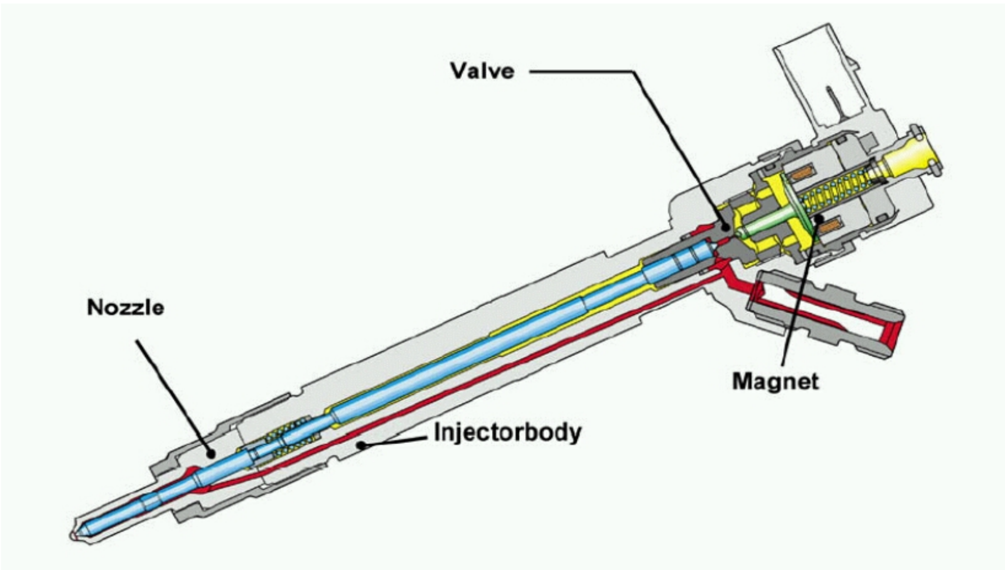


Figure 5-1: Common Rail Injector with a Two-way Solenoid Valve

This injector is too complex for start. Hence we begin modeling with a basic layout of the common rail injector shown in *Figure 5-2*.

Schematic of the injector is depicted in *Figure 5-3* and the corresponding BOOST Hydsim model in *Figure 5-4*. The model consists of the following elements:

- Rail Pressure **(BOUNDARY/Pressure)**
- Injector Tube **(LINE/Laplace Transform)**
- Junction **(JUNCTION/Tee(angle))**
- Holder Bore **(LINE/Laplace Transform)**

- Nozzle Bore **(LINE/Laplace Transform)**
- Nozzle Volume **(VOLUME/Standard)**
- Control Volume **(VOLUME/Standard)**
- Valve Volume **(VOLUME/Standard)**
- Solenoid Valve **(THROTTLE/Time-controlled)**
- Nozzle Orifice **[NOZZLE/VCO (basic model)]**
- Cylinder Pressure **[BOUNDARY/Pressure)**
- Spill Volume **(VOLUME/Standard)**
- Fuel Tank **(BOUNDARY/Pressure)**
- Nozzle Holder **(BOUNDARY/Mechanical)**
- Needle **[NEEDLE/Standard (up-to-date model)]**
- Control Piston **(PISTON/Standard)**
- Piston Leakage **(LEAKAGE/Annular Gap)**
- Nozzle Leakage **(LEAKAGE/Annular Gap)**
- Sump Throttle **(ORIFICE/General)**
- Inlet Throttle **(ORIFICE/General)**
- Outlet Throttle **(ORIFICE/General)**

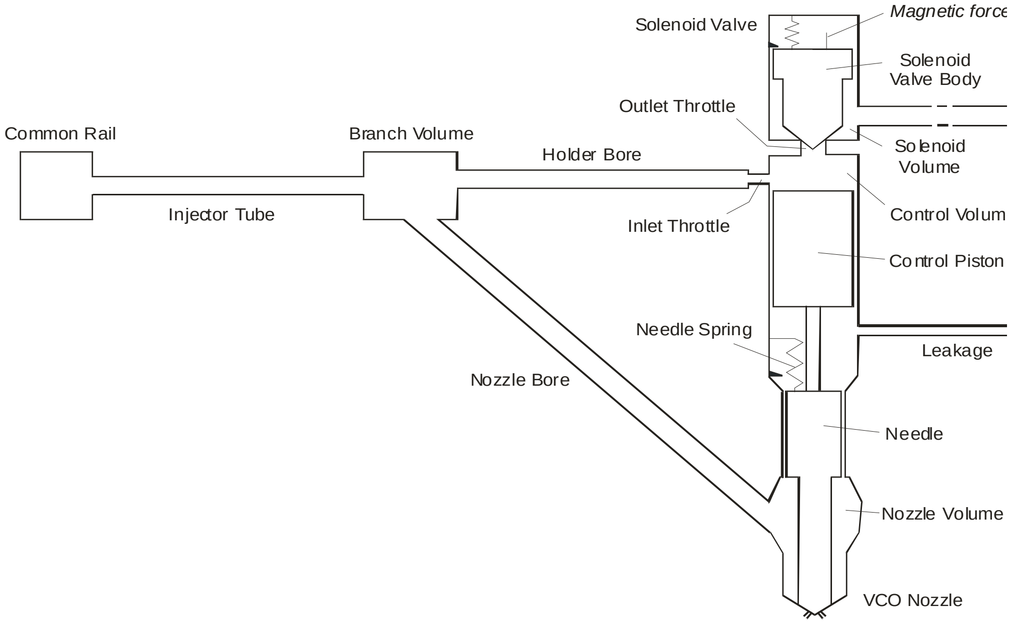


Figure 5-2: Layout of Common Rail Injector with 2/2-Way Solenoid Valve

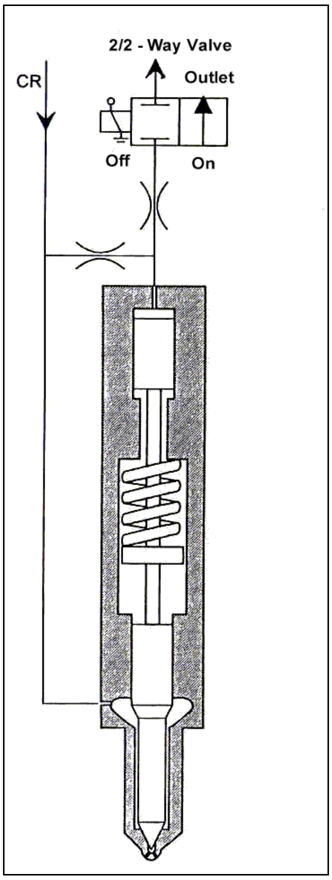


Figure 5-3: Schematic of Common Rail Injector with 2/2-Way Valve

5.1.2. Creating the Model

The system begins at the common rail, which is modeled as a pressure boundary with constant pressure of 1500 bar (Rail Pressure 13[3]). Injector Tube 1 connects the injector with the rail. At this point (Junction 21), the system splits into two branches: one going through Nozzle Bore 4 down to Nozzle Volume 5, and the other via Holder Bore 3 and Inlet Throttle 12 up to Control Volume 6. Nozzle Volume 5 is connected to Needle 18 and Nozzle Orifice 19.

Nozzle Orifice 19 is linked to Cylinder Pressure 14, which is a boundary condition with constant pressure of 100 bar. Control Volume 6 is connected on one side to Control Piston 11 and on the other side to Outlet Throttle 17, which has a larger cross-sectional area than of Inlet Throttle 14. Outlet Throttle 17 is connected via a small Valve Volume 7 to Solenoid Valve 8, which is modeled as a **Time-controlled Throttle** with predefined switching times.

[3] Numbers given for elements may differ from those in your model. Reference *Figure 5-4* for numbered items.

A more complex modeling of the solenoid valve is feasible in BOOST Hydsim as discussed in *Section 5.1.5*. It considers the dynamics of the armature and valve ball but requires much more modeling effort. Solenoid Valve 8 controls the closing/opening of the fuel outlet line, which comprises Spill Volume 9, Sump Throttle 16 and Fuel Tank 15. Back side of Needle 18 and Control Piston 11 are also connected to Fuel Tank 15 (**Pressure Boundary** with a pressure of 1.5 bar).

Up to now, all connections between the icons are hydraulic. There is no data associated with a hydraulic connection (it is only a symbol). In addition to hydraulic connections, Needle 19 also has two mechanical connections: one (relatively soft spring) denoting the nozzle spring and the other (stiff rod) denoting the connection of Needle 19 to Control Piston 11. Both mechanical connections have to be supplied with input data as described in the previous section (refer to *Figure 4-5*).

The needle spring is attached to the fixed support (Nozzle Holder 10), which is modeled as a **Mechanical Boundary** with no motion. The oil leakages through the guide of Needle 18 and through Control Piston 11 are modeled with two leakage elements: Nozzle Leakage 21 and Piston Leakage 20. These icons have the special connections to Needle 18 and Control Piston 11 to identify the “leaking” element to which they belong. Needle 18 has a special connection to Nozzle Orifice 19.

All tubes and drillings of the hydraulic system are modeled as **Laplace Lines**. The Laplace transform calculates the quickest, and therefore is strongly recommended for larger systems as long the line is not too long and the output at line ends is sufficient. If the output (e.g. pressure distribution) along the line is required, the **Characteristics**, **Godunov** or **MacCormack Line** model has to be used. However, these methods require a considerably larger computational effort than the semi-analytical **Laplace transform** solution.

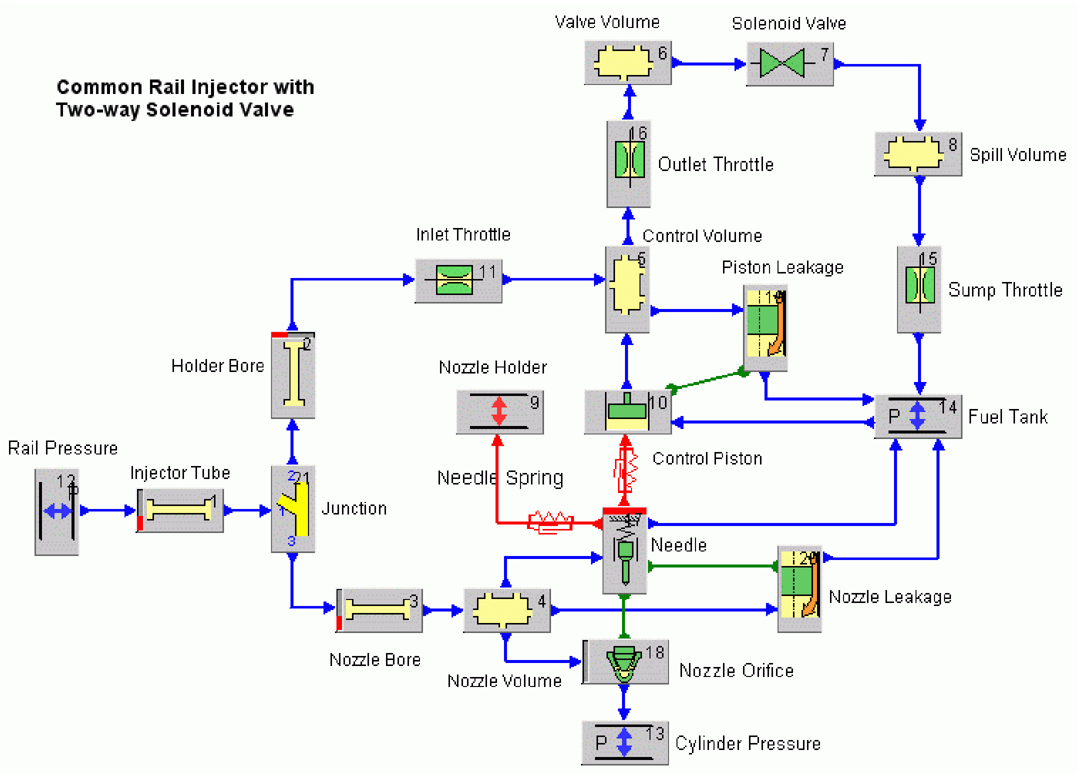
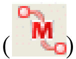
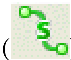


Figure 5-4: BOOST Hydsim Model of Common Rail Injector with 2/2-Way Valve

The basic steps of the model creation are summarized below. Please refer to the example in *Chapter 4* for the basic details.

1. Create a **Boundary Condition** (Rail Pressure). Under **Elements,** expand **BOUNDARY** and double click **Pressure/Temperature** to place the icon in the workspace. Change the default name Pressure to Rail Pressure.
2. Similarly, create the following elements:
   - Injector Tube from **Laplace Transform/LINE** group
   - Junction from **Tee(angle)/JUNCTION** group

   To rotate an element as shown in the example, highlight the element and press “r” on the keyboard. The element will then rotate 90 degrees clockwise.
3. Use the hydraulic connection () to connect Rail Pressure to Injector Tube and Injector Tube to Junction by hydraulic connections (blue lines).
4. Create Nozzle Bore and Holder Bore from **Laplace Transform/LINE** (both use the same type of element for representation in the model).
5. Create the Inlet Throttle and Outlet Throttle from **General/ORIFICE**. Establish the necessary hydraulic connections.
6. Create Control Piston from **Standard/PISTON.**
7. Create Needle from **Standard/NEEDLE**.
8. Create Nozzle Volume and Control Volume from **Standard/VOLUME**.
9. Create Nozzle Orifice from **VCO (Basic model) / NOZZLE** group.
10. Use hydraulic connections to connect Nozzle Volume to Needle and Nozzle Orifice.
11. Connect Needle to Control Piston by a mechanical connection ().
12. Connect Needle to Nozzle Orifice by a special connection ().
13. Create the two boundary condition icons: Cylinder Pressure from **Pressure/BOUNDARY** and Nozzle Holder from **Mechanical/BOUNDARY**.
14. Connect Nozzle Orifice to the Cylinder Pressure by hydraulic connection and Needle to the Nozzle Holder by a mechanical connection (Nozzle Spring).
15. Create Solenoid Valve Volume and Spill Volume from **Standard/VOLUME**.
16. Create Solenoid Valve from **Time-controlled/THROTTLE**.
17. Create Sump Throttle from **General/ORIFICE**.
18. Create Fuel Tank from **Pressure/BOUNDARY**,
19. Join the respective icons (Steps 14 — 18) with hydraulic connections.
20. Create Nozzle Leakage and Piston Leakage from **Annular Gap**/**LEAKAGE**.
21. Use special connections to connect Nozzle Leakage to Needle and Piston Leakage to Control Piston.
22. Establish hydraulic connections from **Control Volume** to Piston Leakage and Nozzle Volume to Nozzle Leakage.

23. Establish hydraulic connections from Needle and Control Piston Leakage elements to the Fuel Tank.

After completing the common rail injector model, it should look similar to *Figure 5-4*. Other than the positioning of the icons, the elements of your model must be strictly the same as in *Figure 5-4* and the connections between the icons and their directions must be maintained. Please check it carefully before you proceed further.

5.1.3. Input Data

1. Specify the input data for each element by double-clicking the icon or clicking on the icon with the right mouse button and select **Properties** from the submenu. The input data for various element types was described in Chapter 4 and the dialogs of new elements will be described here.
2. An expedient way for specifying input data is to begin with the Rail Pressure, a pressure boundary condition element, as shown in the following window:

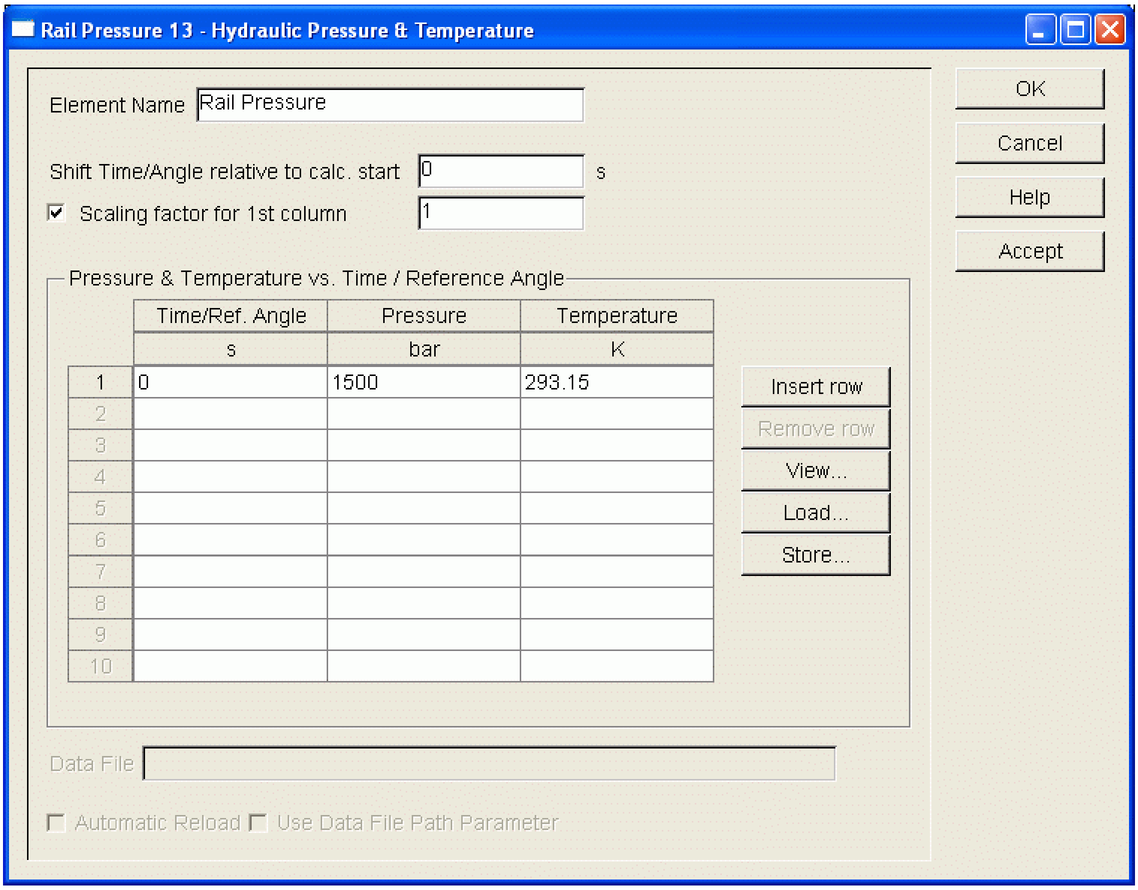


Figure 5-5: Input Dialog of Rail Pressure Boundary

3. Specify the following data for the Cylinder Pressure and Fuel Tank pressure boundary elements:
   - For Cylinder Pressure, set the pressure to 100 bar.
   - For Fuel Tank – 1.5 bar.
4. Data is not required for the Nozzle Holder as it is a fixed mechanical boundary. Change the default name to `Nozzle Holder`.

5. Define the input data for the lines. In the Injector Tube (**Laplace Line**) dialog, specify the line length `200` mm and diameter `2.4` mm. Click **OK** to save the entered data:

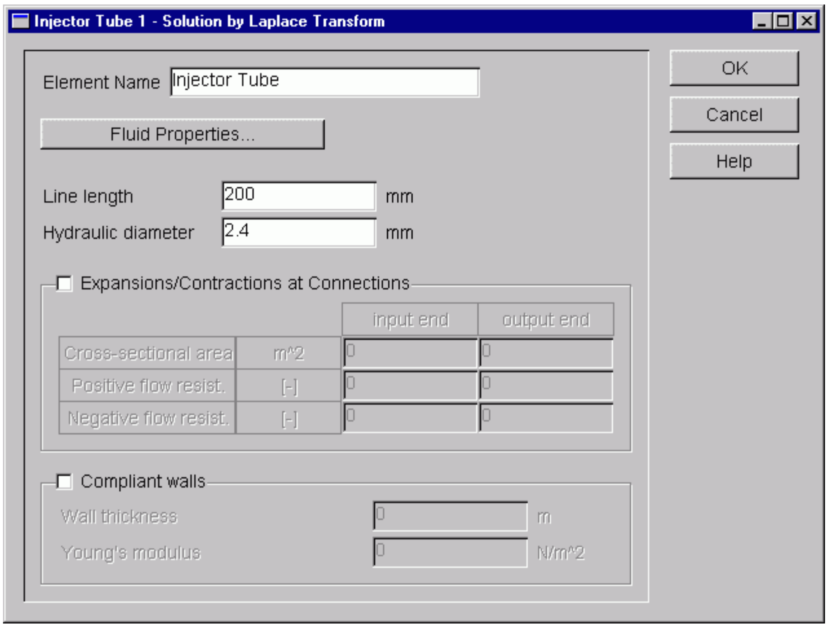


Figure 5-6: Input Dialog of Injector Tube (Laplace Line)

6. Similarly, define the input data for the remaining two lines:
   - Nozzle Bore: length `110` mm and diameter `2.4` mm.
   - Holder Bore: length `70` mm and diameter `2.4` mm.

7. If a line has extensions or contractions at the ends (i.e. a sudden change of cross-sectional flow area), activate **Extension/Contractions at Connections** and specify the necessary cross-sectional areas and flow resistance coefficients.

8. If the change of sound velocity in the line has to be accounted due to the compliance of line walls, activate **Compliant walls** and specify wall thickness and Young's modulus (for more information, refer to the BOOST Hydsim Users Guide).

9. Proceed with specifying the input data for **Volume** elements. The input window of the Nozzle Volume is shown in *Figure 5-7*.

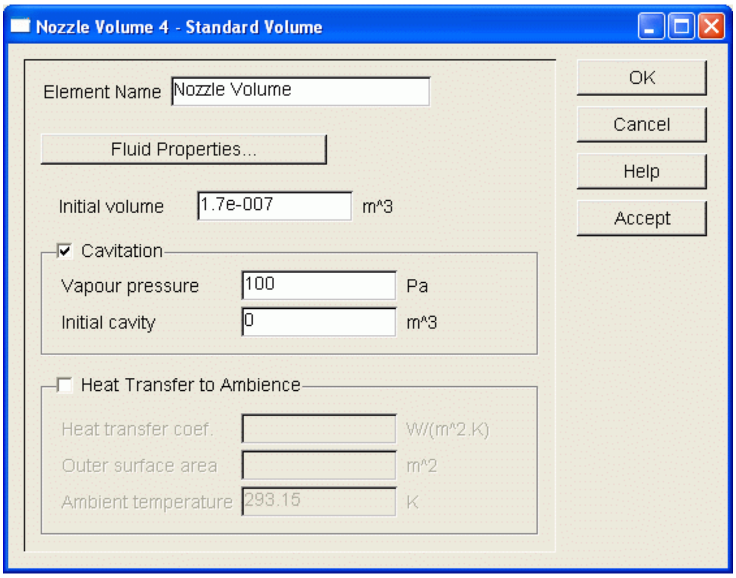


Figure 5-7: Input Dialog of Nozzle Volume

Enter the initial volume `180` mm^3. Keep the default vapor pressure at `100` Pa. Notice that putting zero pressure instead or specifying no value would imply neglecting of the cavitation effects and the possibility of pressure drop below zero.

10. Enter input data (initial volume) for the other volumes:
    - Control Volume: `20` mm^3
    - Valve Seat Volume: `15` mm^3
    - Spill Volume: `300` mm^3

11. In Junction (angle) dialog, angle between Ports 1 and 2 must be specified:

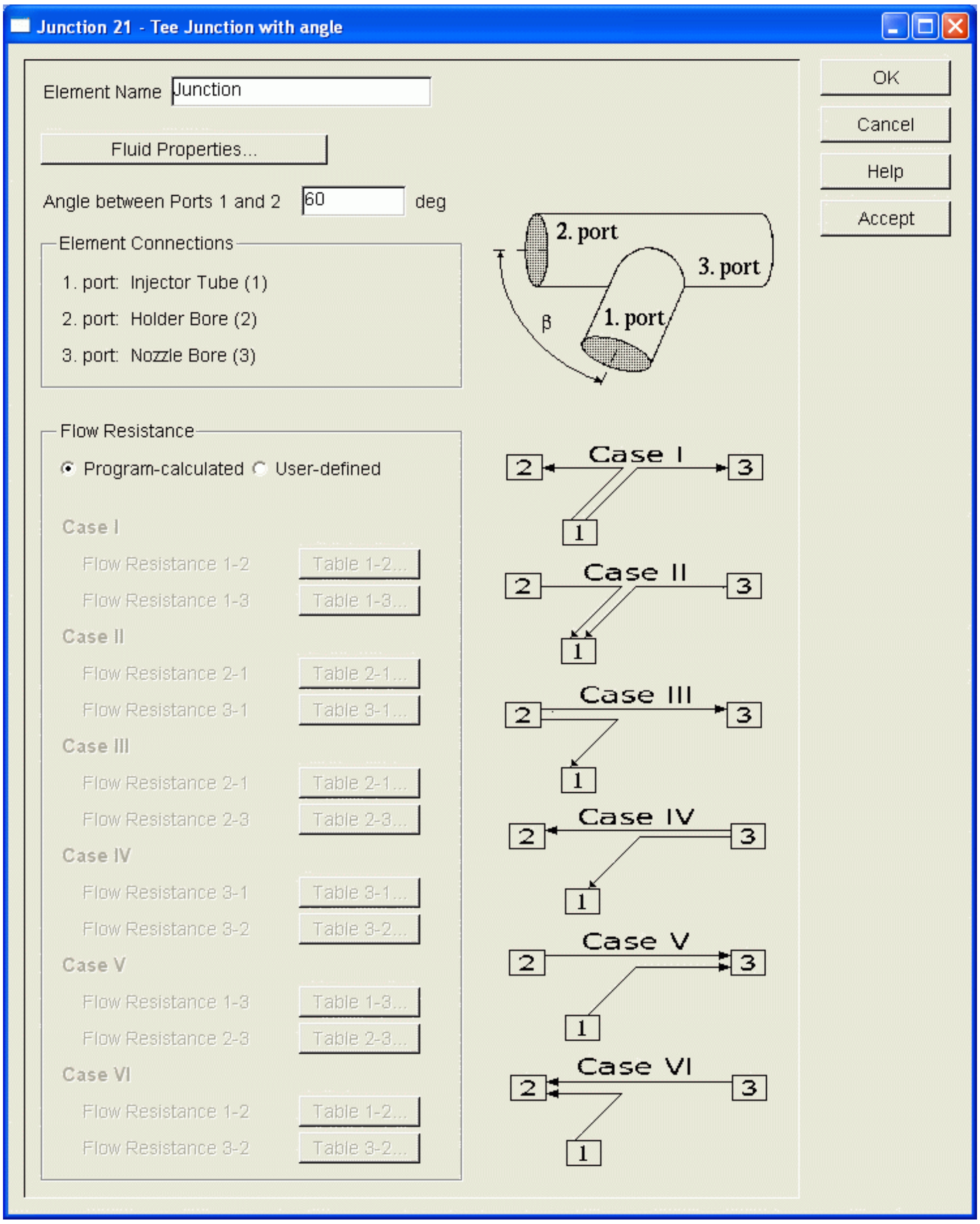


Figure 5-8: Input Dialog of Tee (angle) Junction

12. The Inlet Throttle dialog opens as shown in the following window:

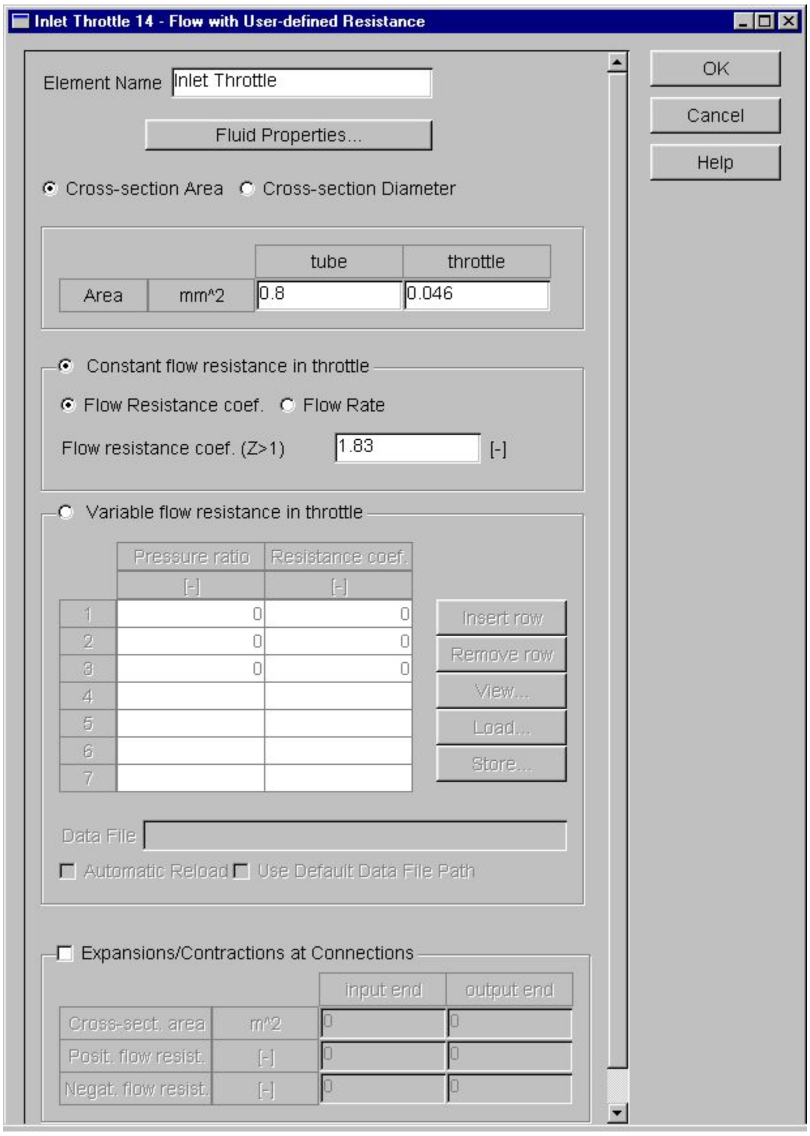


Figure 5-9: Input Dialog of Inlet Throttle

a. Enter the nominal and the narrowest cross-sectional areas of the throttle orifice - `0.80` mm^2 and `0.046` mm^2, respectively.

> ☞ **Note:** Alternatively to the cross-sectional areas, tube and throttle diameters may be specified on input.

b. Specify the flow resistance (loss) coefficient in the throttle equal to 1.83. If the orifice has contractions or expansions at input or output ends, their cross-sectional areas must have corresponding flow resistances (in both directions), and must be specified separately in **Expansions/Contractions at Connections**.

12. Specify the following for the Outlet Throttle:
- Nominal cross-sectional area `0.80` mm^2
- Cross-sectional area of the throttle `0.057` mm^2
- Flow resistance coefficient `1.83`

The Outlet Throttle has a larger cross-section (diameter) than the Inlet Throttle.

13. At this stage enter the input data for the Sump Throttle:
    - Nominal cross-sectional area `2.54` mm²
    - Cross-sectional area of the throttle `0.10` mm²
    - Flow resistance coefficient `1.6`
14. Open the input data window of Needle. The standard nozzle (single-spring) has the following appearance:

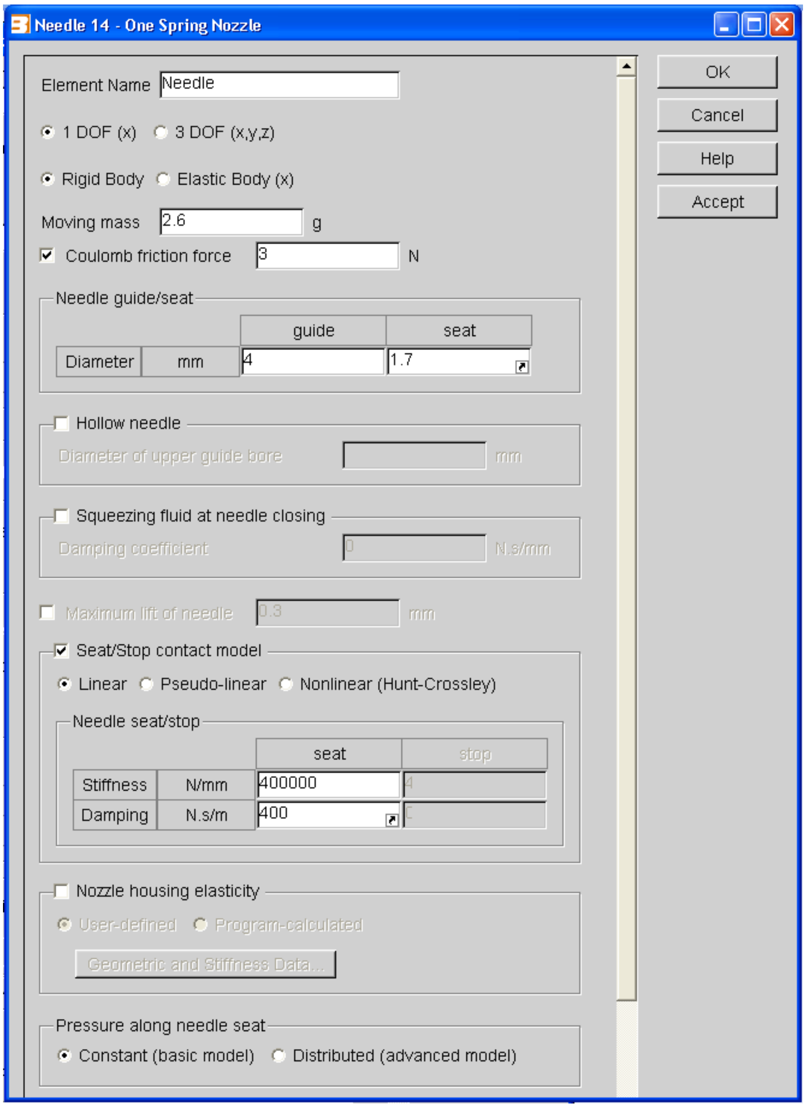


Figure 5-10: Input Dialog of Standard Needle (Up-to-date model)

a) Moving mass: `2.6` g

b) Coulomb friction force: `3` N (optional)

c) Diameter of needle guide: `4` mm

d) Diameter of needle seat: `1.7` mm

e) If any information on the fluid damping at needle closing is available, a viscous damping coefficient of squeezing fluid should be specified. If not, keep the default value (zero). In this situation no additional damping at needle closing will be considered ("dry" needle motion).

f) The stiffness and damping of the needle seat have to be specified. These parameters may not be known exactly but should be set to high, experience-based values. Thus specify `400000` N/mm for the stiffness and `5000` Ns/m for the damping at needle seat. If the Needle has no mechanical stop, deactivate **maximum lift of needle**. In this case, Needle stop data is automatically disabled.

> ☞ **Note:** Lift of needle is limited through Piston stroke in Control Piston 15 element (refer to *Figure 5-14*) which is connected with Needle via stiff mechanical connection.

There are two needle models for standard nozzles in BOOST Hydsim as given in the **NEEDLE** group: **Standard (obsolete model)** and **Standard (up-to-date)**. The input data for both models is the same but the models differ in calculation of the needle opening force (see BOOST Hydsim Users Guide). Basically, the **Obsolete model** (both **standard** and **2-spring**) assumes that at open needle the nozzle volume pressure (i.e. pressure under needle guide) is also acting under the area between the needle seat and nozzle sac (or spray holes inlet). The **Up-to-date model** (**standard** and **2-spring**) is based on the assumption that nozzle sac pressure (or pressure before spray hole inlet) is acting under the above area. For high pressure common rail systems, the difference between nozzle volume pressure and sac/hole inlet pressure at needle opening is very high, therefore the needle opening force and resulting acceleration will differ substantially for both models. Typically, the **Obsolete model** may produce a jump in the needle motion at opening while **Up-to-date model** yields smoother and longer needle opening characteristic. However, the needle acceleration and injection rate might be slightly underestimated in this case. For a common rail application, the **Up-to-date model** has to be used.

The required pre-load of the needle spring for model input data is calculated from the nozzle cracking pressure using the formula:

$$F_{pr} = \frac{\pi\left(d_{guide}^2 - d_{seat}^2\right)}{4} p_{crack} \tag{5.1.1}$$

where d_{guide} and d_{seat} are diameters of needle guide and seat and p_{crack} is the cracking pressure (set nozzle opening pressure). For instance, for d_{guide} = 4 mm, d_{seat} = 2 mm and p_{crack} = 300 bar the calculated nozzle spring pre-load is 282.7 N.

In a real engine, the nozzle will also be affected by the cylinder gas pressure, which is entered in the model as a boundary pressure. This pressure will act on the end of the nozzle needle and have a small effect on assisting the opening of the needle. This can be very important when studying the hydraulic conditions at the end of injection and the possibility for combustion gas blow-back into the nozzle or for secondary injections to occur.

15. Enter input data for Needle Leakage. It is not necessary to specify explicitly the leakage piston diameter as the Leakage element automatically takes the needle guide diameter via the special connection to the Needle element. Leakage gap is pressure-dependent as you may observe from the "User-defined annular gap" table:

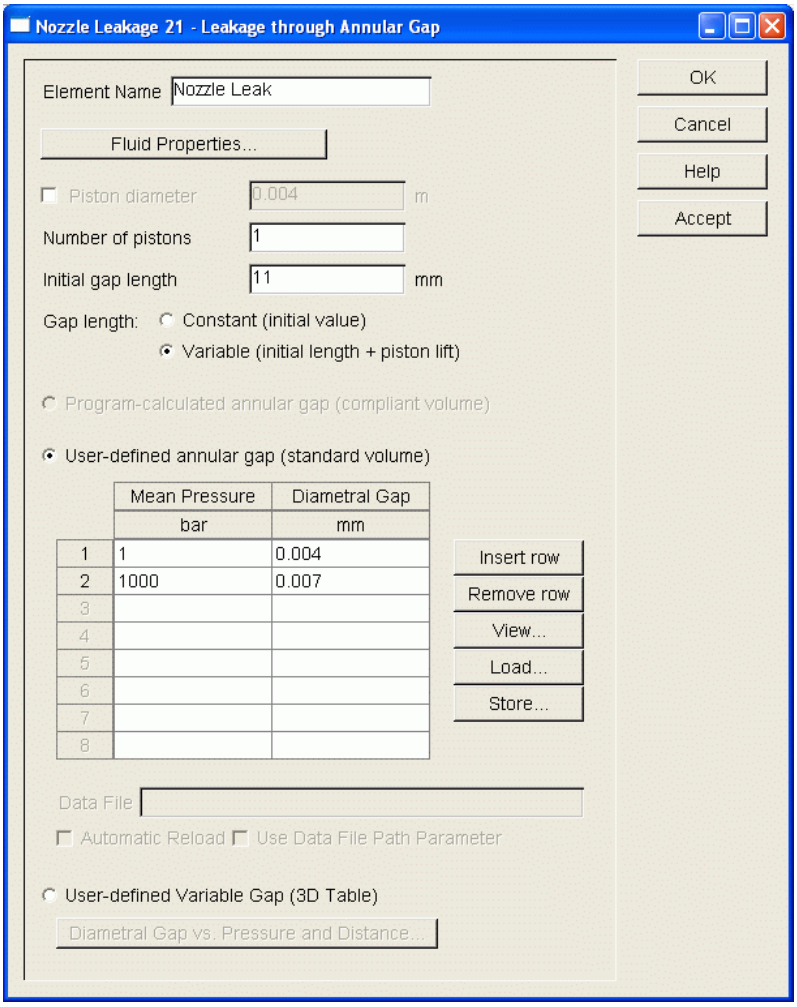


Figure 5-11: Input Dialog of Needle Guide Leakage

16. The Nozzle Orifice dialog opens as follows:

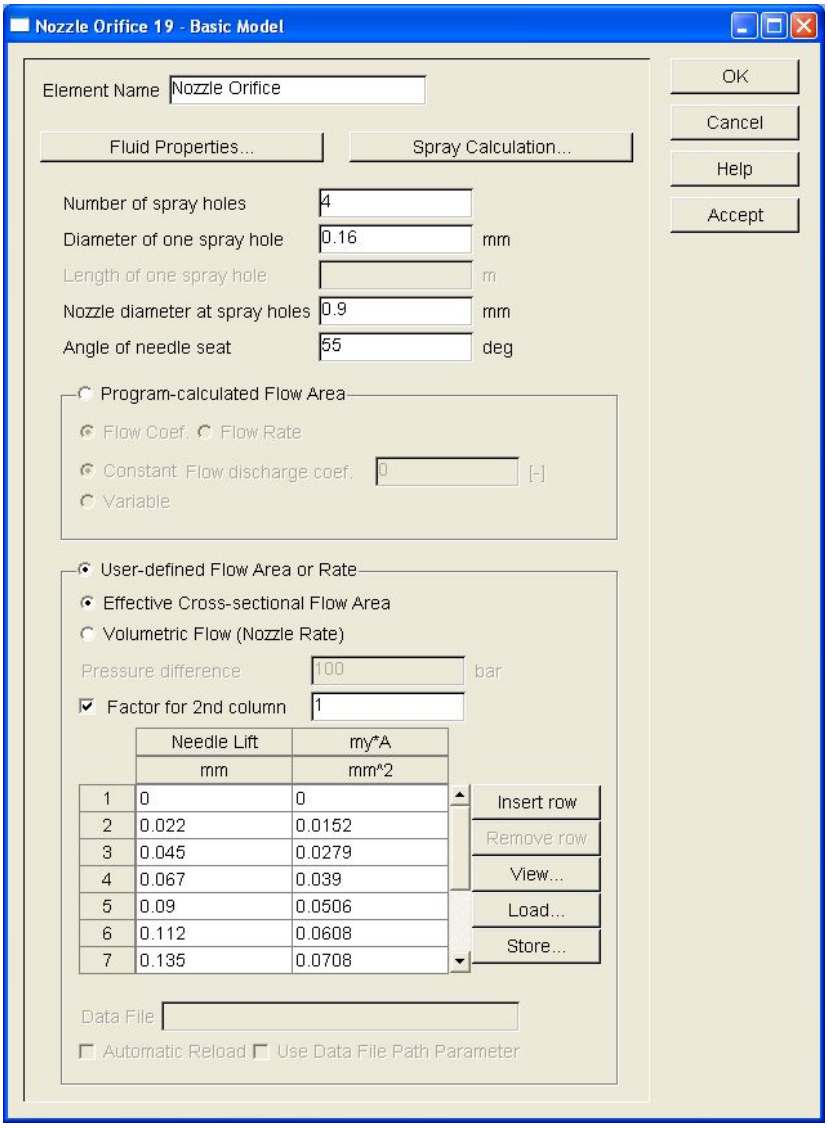


Figure 5-12:) Input Dialog of Basic Nozzle Orifice (VCO)

The two options are:

a) Activate **Program-calculated Flow Area**. The internal BOOST Hydsim algorithm (refer to the BOOST Hydsim Users Guide) can be used for calculating the effective cross-sectional flow area of the nozzle orifice. This possibility is not used here, but may be performed when this exercise is completed. This algorithm assumes that flow discharge coefficient is user-defined. It may be constant or a function of pressure ratio across nozzle. Alternatively to discharge coefficient, the user may input the nozzle flow rate measured at maximum needle lift and respective pressure difference. According to these values, flow discharge coefficient will be calculated and displayed automatically. In addition, the angle of the needle seat (60 degrees), number of spray holes (4) and diameter of one spray hole (0.16 mm) have to be specified. For a more refined modeling of the nozzle orifice, use **VCO orifice (Extended model)**.

b) An alternative is to provide directly, if known, the effective cross-sectional flow area or rate (e.g. calculated externally or measured) vs. needle lift. Activate **User-defined Flow Area or Rate** and **Effective Cross-sectional Flow Area**. The table below will be activated and the user can type in the effective flow area or volumetric rate as a function of needle lift or read the data from an external ASCII file. In this example, the data is already prepared. In this particular BOOST Hydsim model of VCO orifice, if the discharge coefficient is constant (usual case), we cannot distinguish between turbulent and cavitating flow. Therefore this nozzle orifice element is entitled **Basic Model**. However, for the **User-defined Flow Area or Rate** the cavitation effects might be "hidden" in the flow area provided. Hence, this case completely depends upon the user input. To encounter cavitation and other effects by BOOST Hydsim internal model, the current element has to be replaced by another, much more elaborate nozzle orifice element **VCO(extended)** available from **NOZZLE** group. Extended nozzle orifice modules with cavitation are discussed in *Chapter 6*.

c) In the current model, the table `Effective Flow area vs. Needle Lift` is specified. For the actual table data the curve comes to a horizontal line at 0.178 mm lift area (no throttling at seat anymore). Select **View** to display the graph shown on *Figure 5-13*.

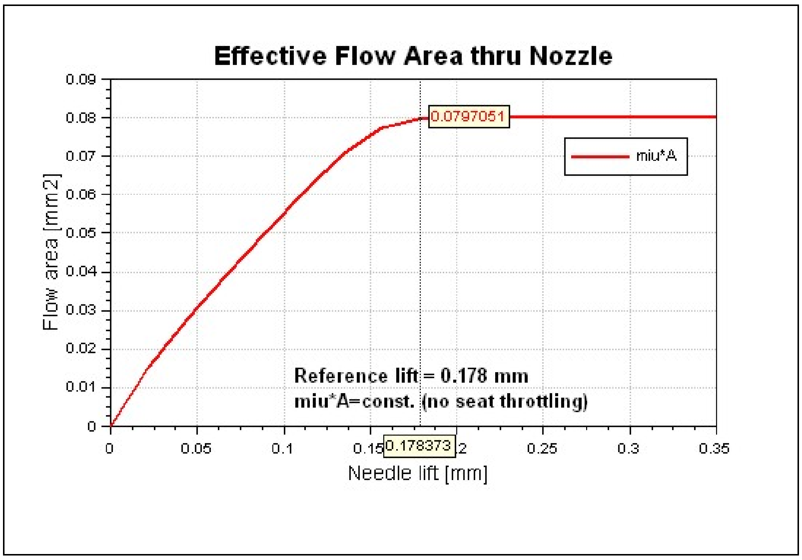


Figure 5-13: Effective Flow Area vs. Needle Lift

17. Define the input data for the Control Piston. This element was previously shown in Chapter 4. Open the input window of Control Piston:

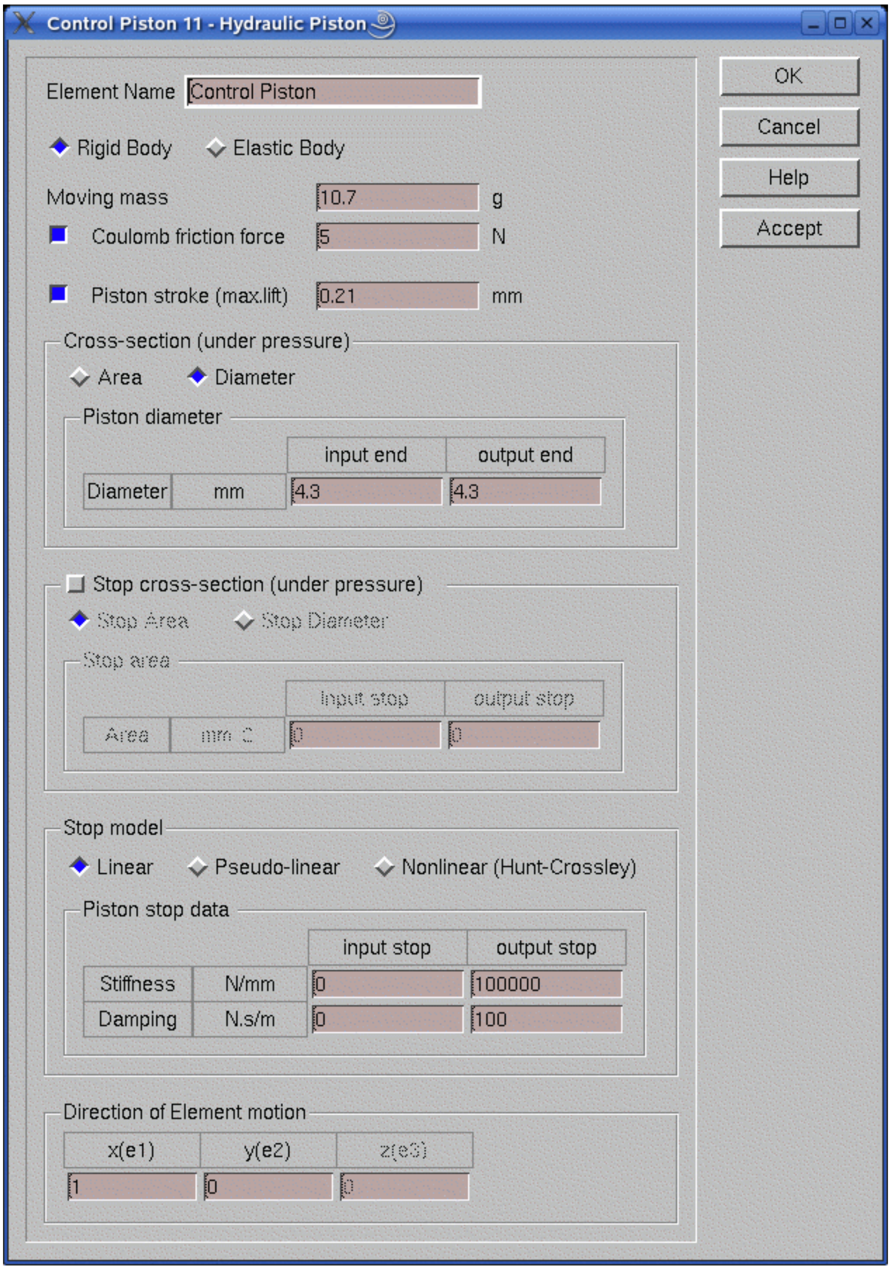


Figure 5-14: Input Data Dialog of Control Piston

The Piston has no separate stop at input end. Its motion downwards is limited by the needle lift due to a very stiff connection between the piston and needle.

18. In this example, the Solenoid Valve is modeled as a variable throttle controlled by the timing of cross-sectional areas (Switch Valve). Open the input dialog of the Solenoid Valve:

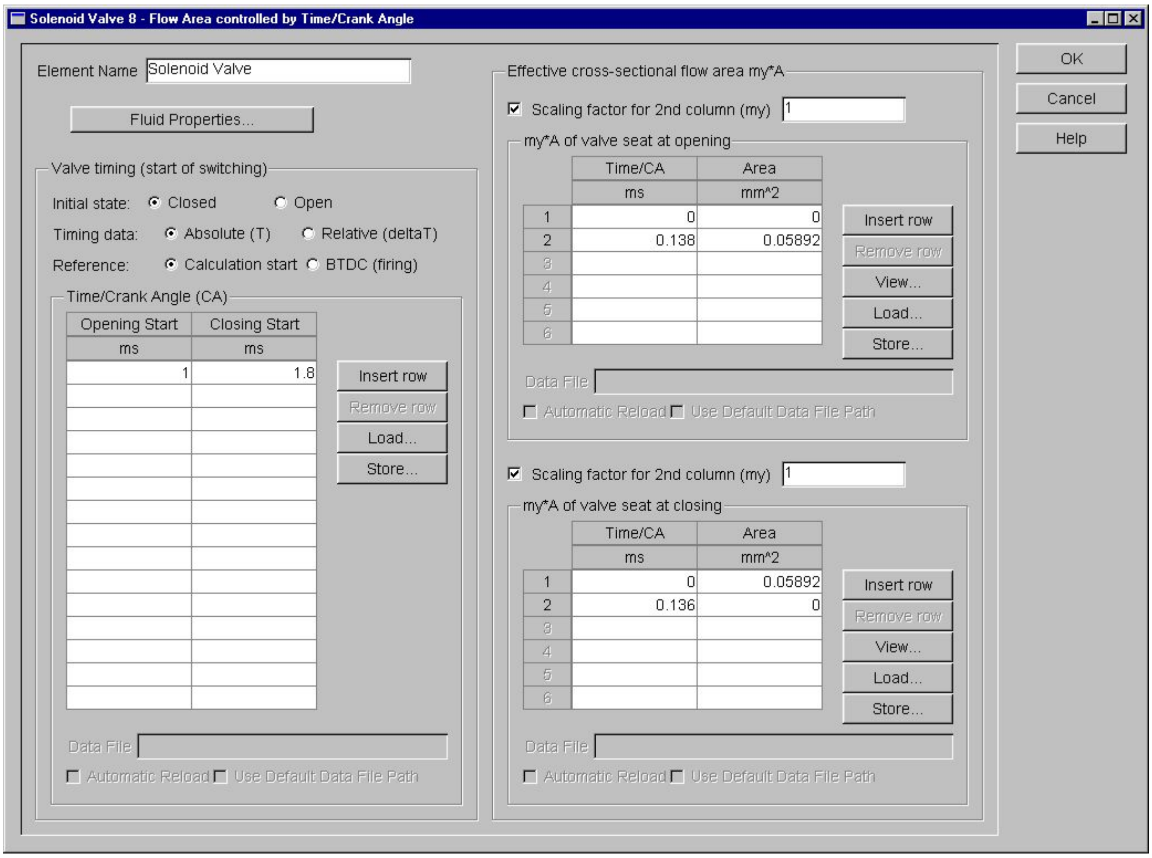


Figure 5-15: Input Data Dialog of Solenoid Valve (Time-controlled Throttle)

18. In the Valve timing field, first define the Initial state of valve, which may be Closed (valve closed) or Open (valve fully opened). Depending on the chosen initial state different column order for Time/Crank angle table will appear. The timing events can be defined in Absolute or Relative domain. For Relative domain each following timing instant refers to the previous event. Absolute timing can refer to the Calculation start or the time before top dead center of firing (BTDC).

19. In the table below, the Time/angle at which the valve opening or closing events start have to be entered. In the two tables beneath the effective cross-sectional flow areas of seat at valve opening and closing time functions have to be provided. The values in between are linearly interpolated.

    Notice that initial flow area will be taken from Effective cross-sectional flow area field according to initial state (closed/opened). Clearly, flow areas of the fully open valve at the end of valve opening and start of closing must be equal. The first zero value (empty cell) implies the end of valve switching procedure (all further values will be ignored).

☞ **Note:** In this model, the solenoid valve is represented by a **Switch Valve** with predefined opening/closing characteristics. The influence of the system dynamics on the valve body motion is neglected. If the dynamics of the solenoid valve has to be considered, the simple **Switch Valve** model must be replaced by a combination of the **Armature** and **Slide Valve**. Armature (Basic or Extended model) is accessible from **SOLENOID** group. Slide Valve (lift-controlled throttle) is available from **THROTTLE** group.

20. All input data is defined for every element of our injection system. It is still necessary to specify the initial conditions. To keep the system equilibrium at the start, ensure the following values are input:
    a) Specify the rail pressure 1500 bar as the initial condition for all volume elements except **Spill Volume** for which an initial condition of 1.5 bar (pressure in Fuel Tank) is required. Use **Edit** | **Initial Conditions** | **Volume** for automatic definition of initial pressure for <u>all</u> volumes (select volume elements before setting 1500 bar).
    b) Specify a negative initial displacement of **Control Piston** -0.088 mm (calculated from the equilibrium of forces on **Piston** and **Needle**). Alternatively, a preload of 2176 N for needle-piston connection spring (rod) can be specified. If the preload or initial condition is not defined (zero by default), the system equilibrium will be reached after a certain period of time. However, initial pressures in the high-pressure volumes have to be specified correctly. Otherwise high flow oscillations will occur in the system and the calculation results will be completely different than in the case of starting from dynamic equilibrium (the stationary state will not be reached before injection start).
21. At this stage, proceed with the selection of the output parameters. The output parameter list is available from the **Element | Store Results** menu for each element. A simpler way to access the window is to open the **Element** pull-down menu and click **Store Results**. Typical output parameters for a common rail injector would be pressures in the volumes, motion of needle and piston, volumetric flow rates through inlet/outlet throttles and solenoid valve, injection rate, total amount of injected fuel, leakage through needle guide and piston, etc. Of course, the user is free to choose any output parameters he/she prefers. The only recommendation is to avoid choosing too many output parameters, as this may require a large amount of disk space.
22. **Simulation** | **Control**. Specify the following:
    a. Domain: Time[s]
    b. Calculation step: 2e-07
    c. Time interval: 0.003
    d. Number of values to be stored: 300
23. **Model** | **Fluid Properties**. Assume diesel for hydraulic fluid and define the following under **Constant Fluid Properties**:
    a. Bulk modulus]N/mm^2]: 1630
    b. Fluid density [kg/m^3]: 865
    c. Kinematic viscosity(cSt): 1.7

The user can specify different constants or variable fluid properties if necessary. Definition of variable (pressure-dependent) properties will yield more accurate calculation results and is highly recommended for practical applications. These need not be complex tables to yield good results.

5.1.4. Running Calculation

Refer to *Section 4.4*.

5.1.5. Calculation Results

Refer to *Section 4.5* for inputting results data in the **IMPRESS Chart** screen.

Select **Simulation | Show Results** to open Impress Chart with the calculation results tree (as shown in *Figure 5-16)*. Alternatively load the `results.ppd` file from *…/boost_hyd/common_rail2wv.Case_1* directory.

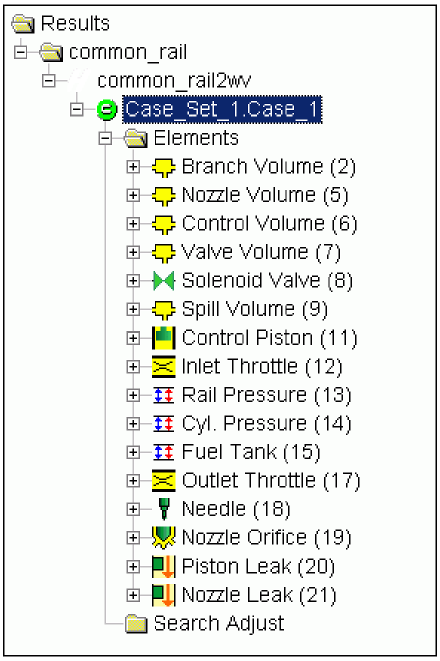


Figure 5-16: Impress Chart Output Tree (Show Results)

Example: User-Defined Cross-Sectional Flow Area

Figure 5-17 shows calculation results for the case with user-defined cross-sectional flow area. The top graph of *Figure 5-17* depicts the **Rail pressure** (1500 bar), pressure variation in the **Junction** then in the **Solenoid, Control** and **Nozzle Volumes** and inlet pressure (pressure before spray holes) of **Nozzle Orifice**. At the beginning, the solenoid valve is closed and pressure in all volumes stays constant at 1500 bar. At the opening of the solenoid valve, the pressure in the **Control Volume** drops rapidly down as the flow rate through **Outlet Throttle** is higher than the flow rate through **Inlet Throttle** as shown in the *Figure 5-17*, bottom graph. Remember that the cross-sectional area of the **Outlet Throttle** is greater than the area of the **Inlet Throttle**. Pressure in **Junction** and **Nozzle Volumes** starts dropping after a certain time delay at a relatively slow rate.

As these volumes are constantly filled in from the common rail, their pressure rises up again after some time which depends on the length of **Injector Tube** and **Nozzle Bore** and the velocity of sound. In fact, pressure in these volumes oscillates around the common rail pressure.

a) As soon as the pressure difference between **Nozzle** and **Control Volumes** reaches nozzle cracking (opening) pressure, both **Needle** and **Control Piston** lift up (as shown in *Figure 5-18*, top graph) and the injection process begins. The injection rate is plotted in *Figure 5-18*, middle graph. Notice that **Control Piston** motion is limited by its stroke (0.21 mm), therefore both **Piston** and **Needle** stay at the limiting stop (top position) during the injection. The injection duration is controlled by the solenoid valve and depends on the actual engine load.

b) When the solenoid valve starts closing, **Outlet Throttle** closes and the pressure in **Control Volume** rapidly rises. Fuel flows through the **Inlet Throttle** into the **Control Volume** as shown in *Figure 5-17* (refer to curve Inlet Throttle: volum. rate). Shortly, the pressure difference between **Nozzle** and **Control Volumes** drops down to the nozzle closing pressure and **Needle** closes. The injection process is then over.

c) *Figure 5-18*, bottom graph, shows the leakage rate through **Needle** guide and **Control Piston**. The user may observe that this leakage is quite small compared to the injection rate (middle graph). However, it can be quite significant in common rail systems because it occurs continuously, not just during injection, and should be checked during modeling.

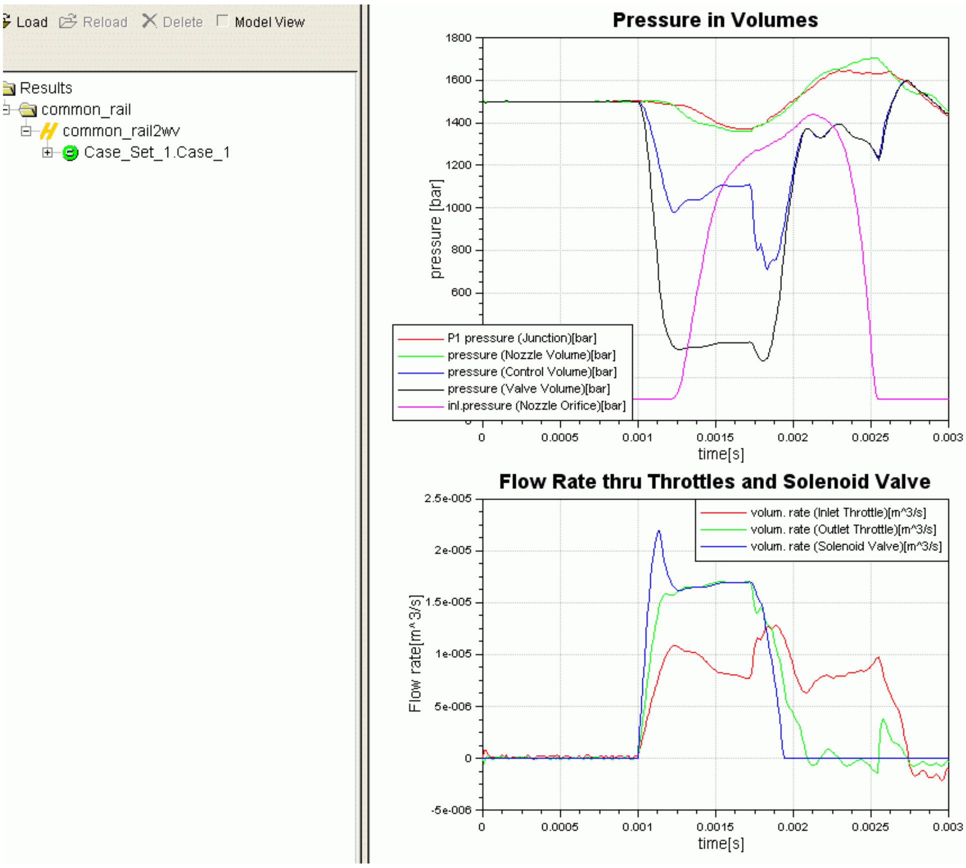


Figure 5-17: Pressure in Junction and Volumes and Flow Rates through Inlet/Outlet Throttles and Solenoid Valve

☞ **Note:** While modeling leakage through pressure-dependent annular gap, it is necessary to specify the annular gap at a peak pressure somewhat higher than the expected peak pressure at the current point. Knowing the actual pressure, BOOST Hydsim interpolates between the annular gap values. Note that BOOST Hydsim uses the average (mean) pressure between the two ends (e.g. volumes) of the leakage path for the leakage rate calculation. Alternatively, BOOST Hydsim can calculate the variable annular gap internally. In this case, the **Leakage** element must be connected to **Compliant Volume** with an active option Compliant Walls. Such a case is shown in the BOOST Hydsim demo-example leakage_gap_comp.hyd.

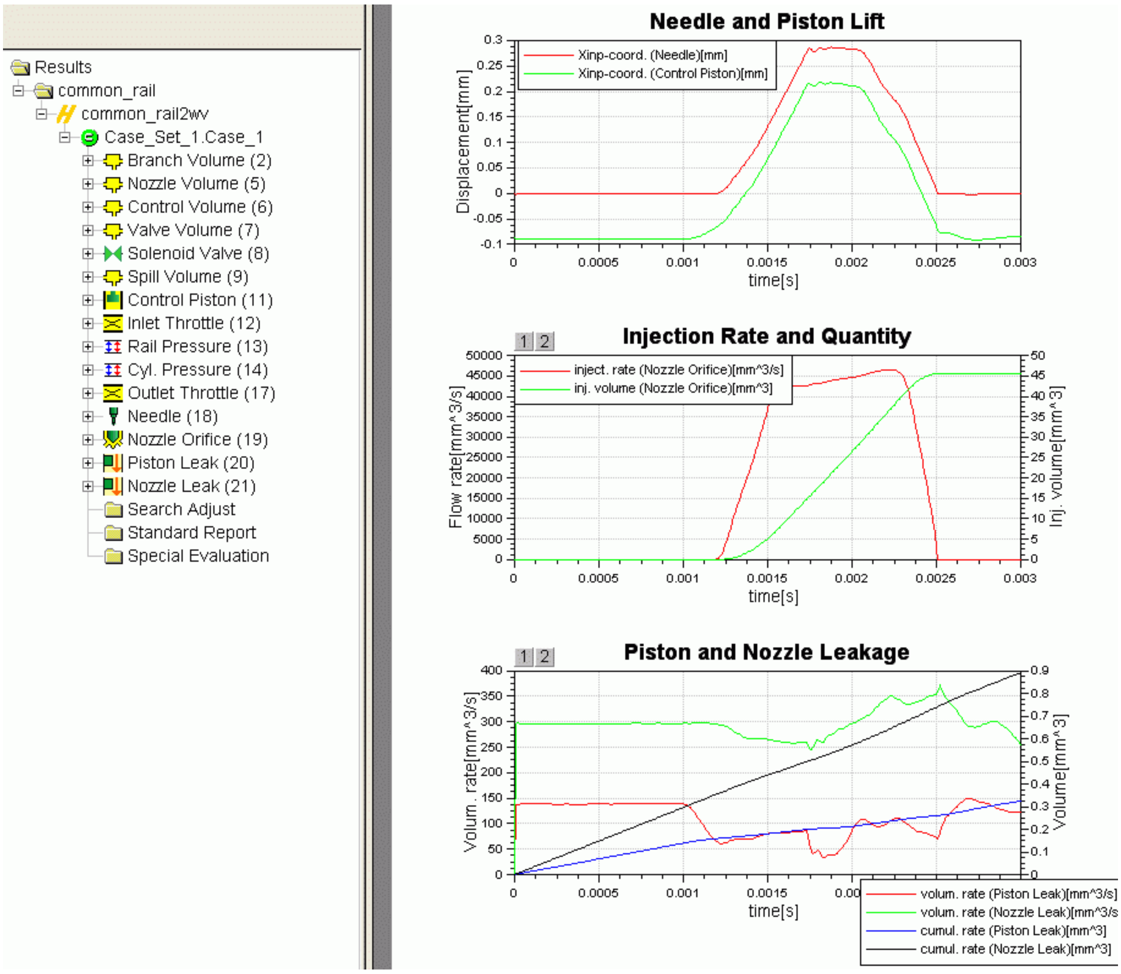


Figure 5-18: Needle and Piston Motion, Fuel Injection Rate, Quantity and Leakage

5.1.6. Animation

To perform animation of injector flow, select **Simulation | Animation ►| Nozzle Flow** and click the check button of the Nozzle Orifice element as shown in *Figure 5-19*.

Figure 5-19: Nozzle Animation Dialog

To animate flow in the nozzle only (refer to *Figure 6-48*), click **OK** to confirm selection. However, for a standard common rail injector with 2/2-way valve an extended animation is available under **Common Rail with 2/2 Way Valve** option. Click 

to open the window shown in *Figure 5-20*. Specify key elements of the injector model: Inlet Throttle, Outlet Throttle and Control Valve. Based on the selected elements, the extended animation model will be created (refer to Figure 5-25).

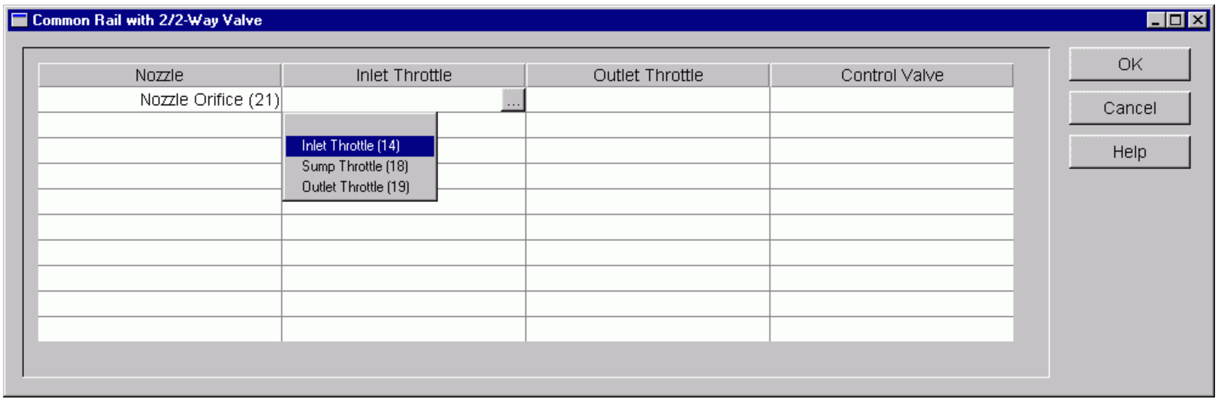


Figure 5-20: Animation Dialog for Common Rail Injector

To get the list of appropriate elements, click in the input field. Then select the right element from the pull-down menu:

> **Note:** Program will collect all Orifice group elements from BOOST Hydsim model in the PullDown menus for Inlet Throttle and Outlet Throttle. For Control Valve, program will collect all Switch Valve, Flow Area vs. Time/CA, Solenoid Armature, Lift Function and Stack Actuator elements.
>
> **Note:** Each selected element will provide an extension in the animation model (refer to Figure 5-25). The user needs not to select them all. However, the elements have to be selected in the successive order from Nozzle to Control Valve, otherwise BOOST Hydsim will issue an error message.

In this example all elements are selected (all columns filled) as shown below:

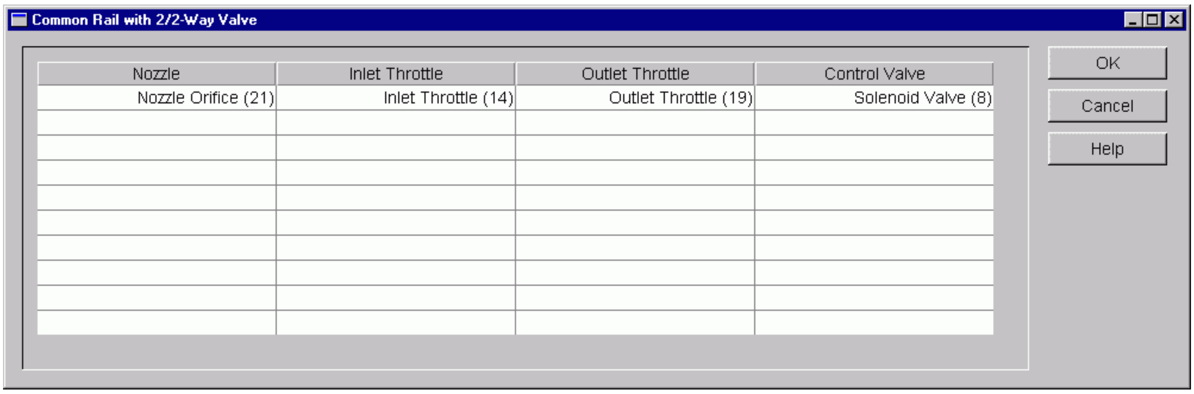


Figure 5-21: Selected Elements in Animation Dialog Box

The animation data file will be stored in case-specific subdirectories with extension `.mod`:
<project_dir>/BOOST Hydsim
/<model_name>.<case_name>/animation/common_rail2wv_nozzle_19.mod.

To enable animation of nozzle spray, spray parameters have to be specified in the Input Data dialog of **Nozzle Orifice/Spray Calculation**.

Select [Spray Calculation......] to open the following:

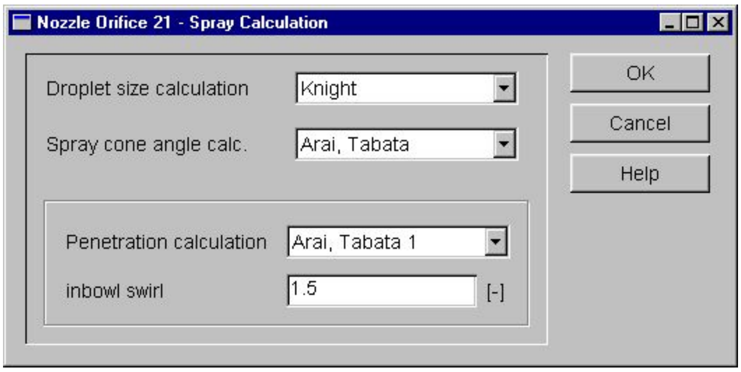


Figure 5-22: Spray Calculation Dialog

From the pull-down menus, select the appropriate empirical algorithms (formulas) for the calculation of the mean droplet size, spray cone angle and penetration.

5.1.7. Spray Calculation Results

Spray calculation results can be plotted in the Impress Chart window (refer to *Figure 5-24*). Activate the Spray parameters toggle switches (Sauter mean diameter, spray cone angle, spray penetration) in the Output Parameters dialog box of Nozzle Orifice, as shown in the following figure:

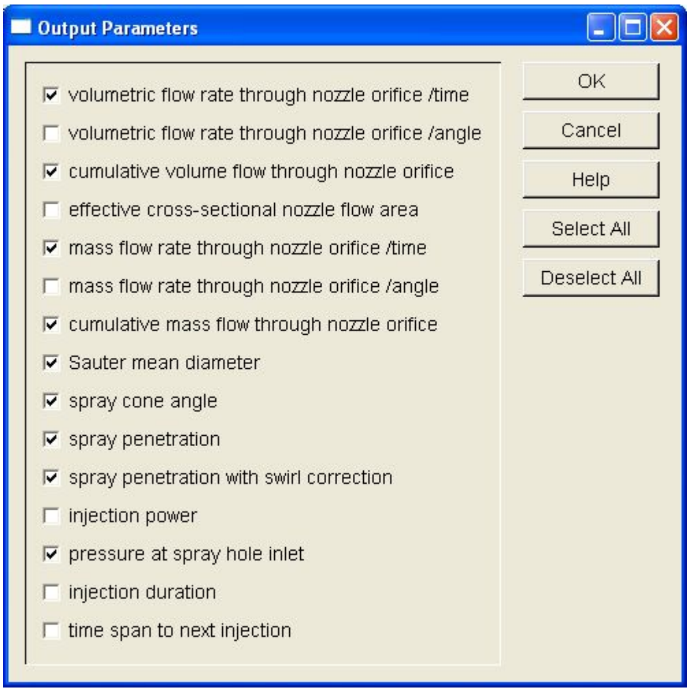


Figure 5-23: Output Data of Nozzle Orifice with Spray Calculation

Refer to *Section 4.5* for loading and viewing results in the Impress Chart window.

Figure 5-24 (top and middle graph) shows the results of spray calculation and pressure before nozzle spray hole inlet (bottom graph).

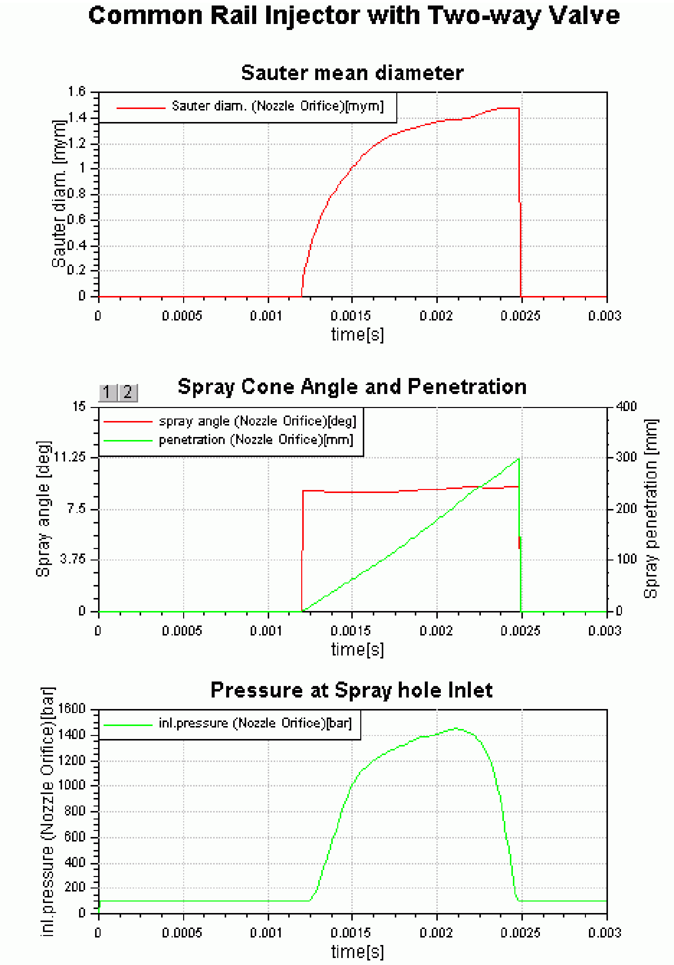


Figure 5-24: Spray Parameters and Injection Pressure

For a detailed description of Spray Calculation refer to the BOOST Hydsim Users Guide.

5.1.8. Running Animation

To perform animation, select 

in the Animation Dialog (refer to *Figure 5-19*) to open the PP3 window and loads the animation file for the actual (previously calculated) model as shown in *Figure 5-25*.

Another, more lengthy way to perform animation is to open explicitly the postprocessor PP3 session and manually load the animation file `common_rail2wv_nozzle_19.mod` with **Open** command or pressing  (as shown in the following figure).

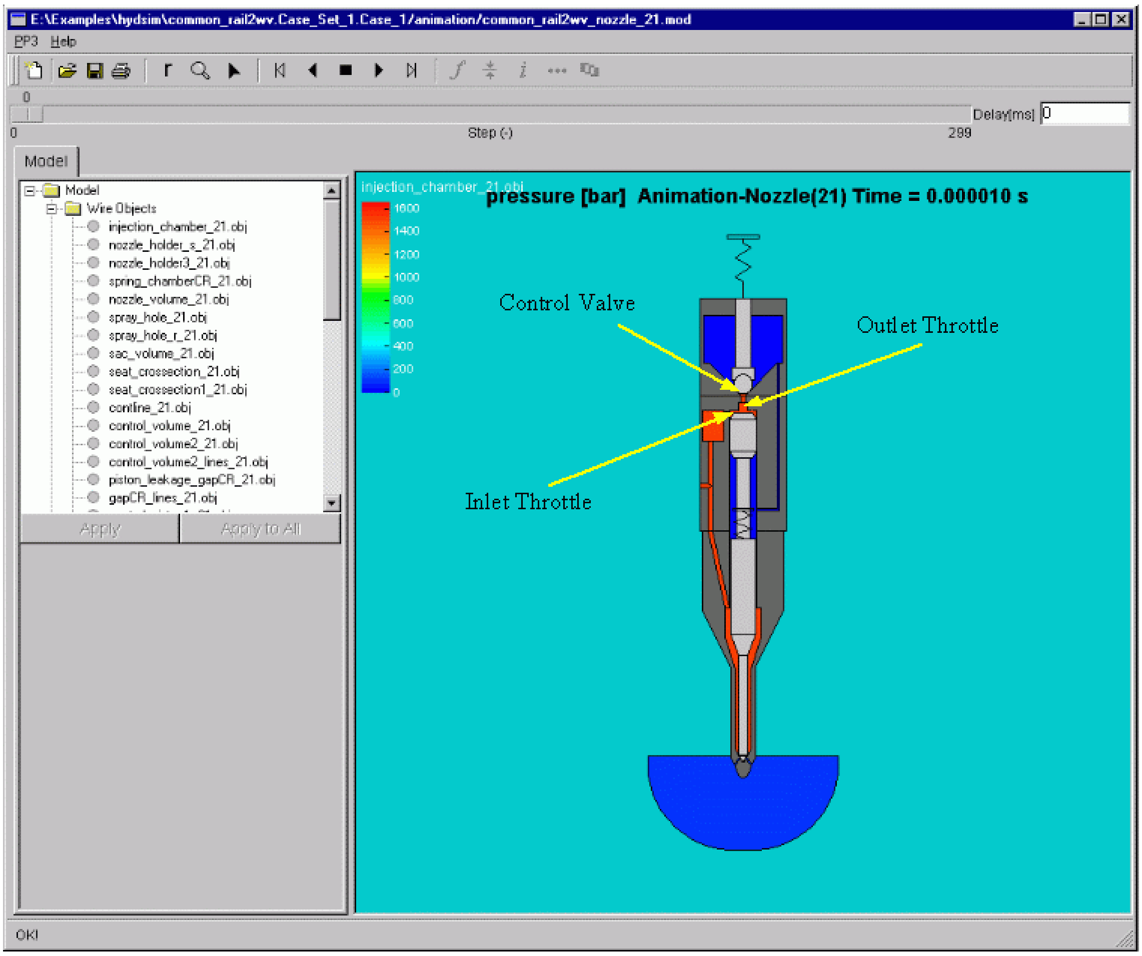
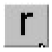
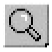
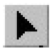
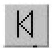
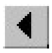
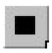
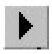
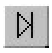


Figure 5-25: PP3 Window with Loaded Animation File

The following buttons are available for animation control:

1.  : Reset
2.  : Zoom
3.  : Change
4.  : Step Backward
5.  : Play Backward
6.  : Stop Animation
7.  : Play Forward (**Start Animation**)
8.  : Step Forward

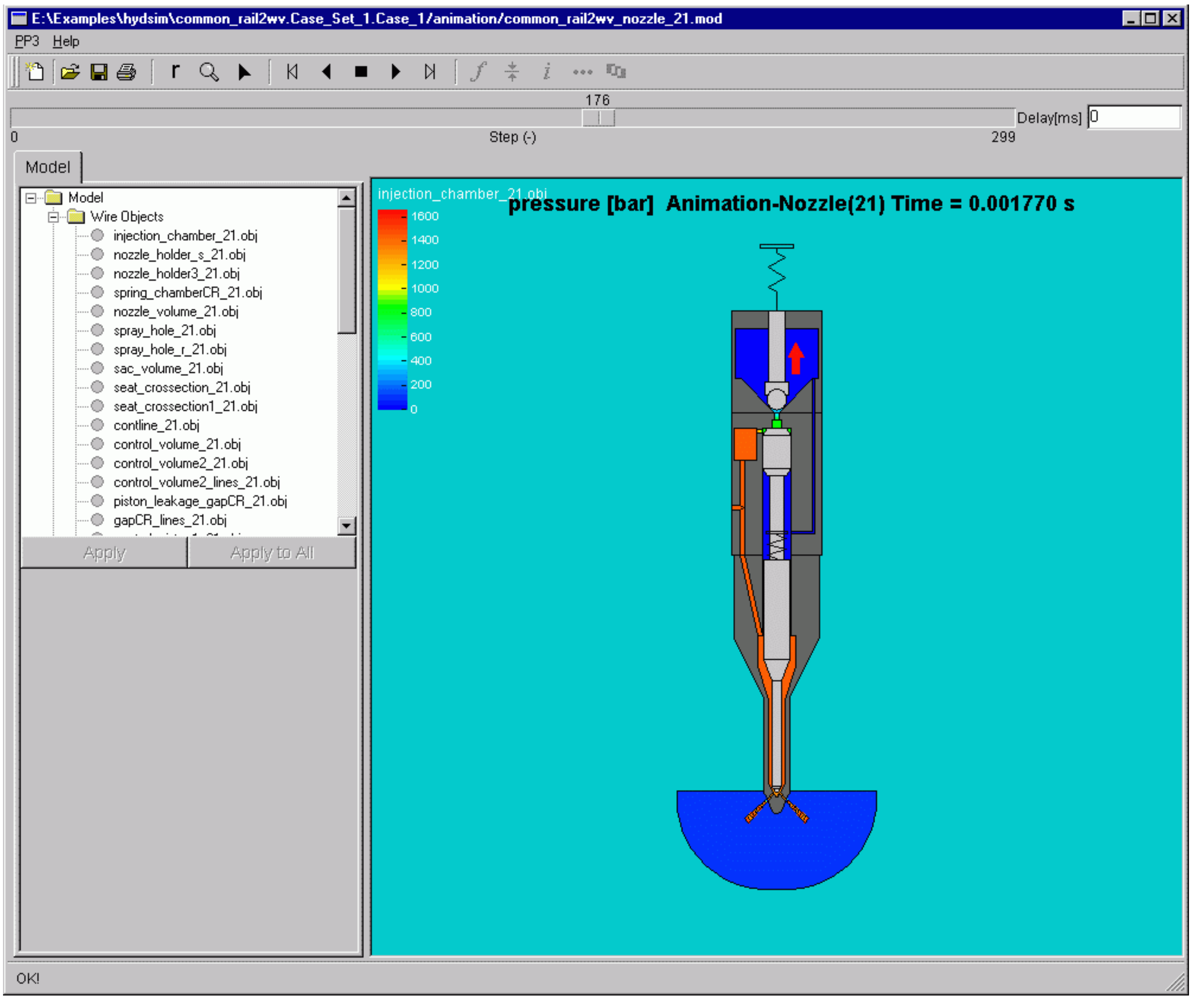


Figure 5-26: PP3 Window with Common Rail Injector Animation

Thick red arrow indicates the actuation of the Solenoid Valve.

Refer to *PP3* section of the GUI Users Guide for more information.

5.2. Injector with a Three-way Solenoid Valve

In this section, a more complex BOOST Hydsim model of a common rail injector is introduced. The injector is controlled by a three-way solenoid valve. The layout of the injector is shown in *Figure 5-27* (with closed needle valve) and *Figure 5-28* (with opened needle valve). Injector schematic is provided in *Figure 5-29*. This model does not represent the design of the real injector: it is just an equivalent dynamic model. Obviously, the injector layout is somewhat similar to the previous one, shown in *Figure 5-2*. The basic difference is the three-way solenoid valve used instead of the two-way valve described previously. Hence the BOOST Hydsim model of the common rail injector in *Figure 5-2* can be taken as a basis for building-up of the actual model. Basically, the model of the solenoid valve and the connecting elements have to be changed. Moreover, another nozzle orifice type will be used: SAC orifice (basic model). Furthermore, the dynamic behavior of the solenoid armature and valve body is considered in this model.

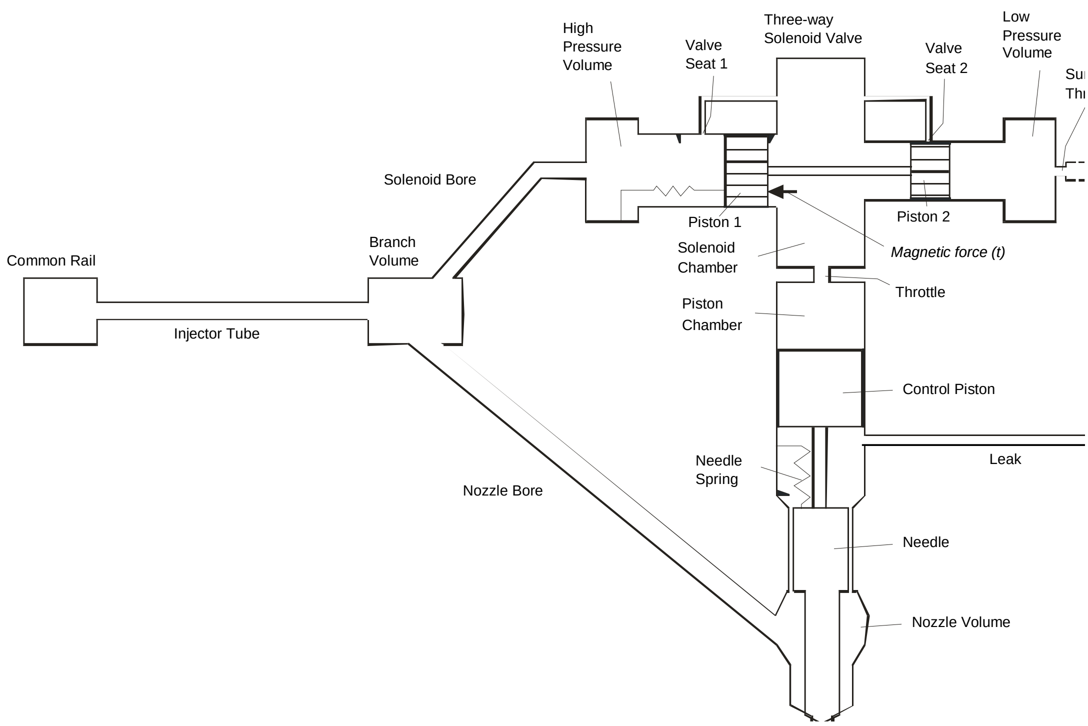


Figure 5-27: Layout of Common Rail Injector with 3/2-Way Valve (needle closed)

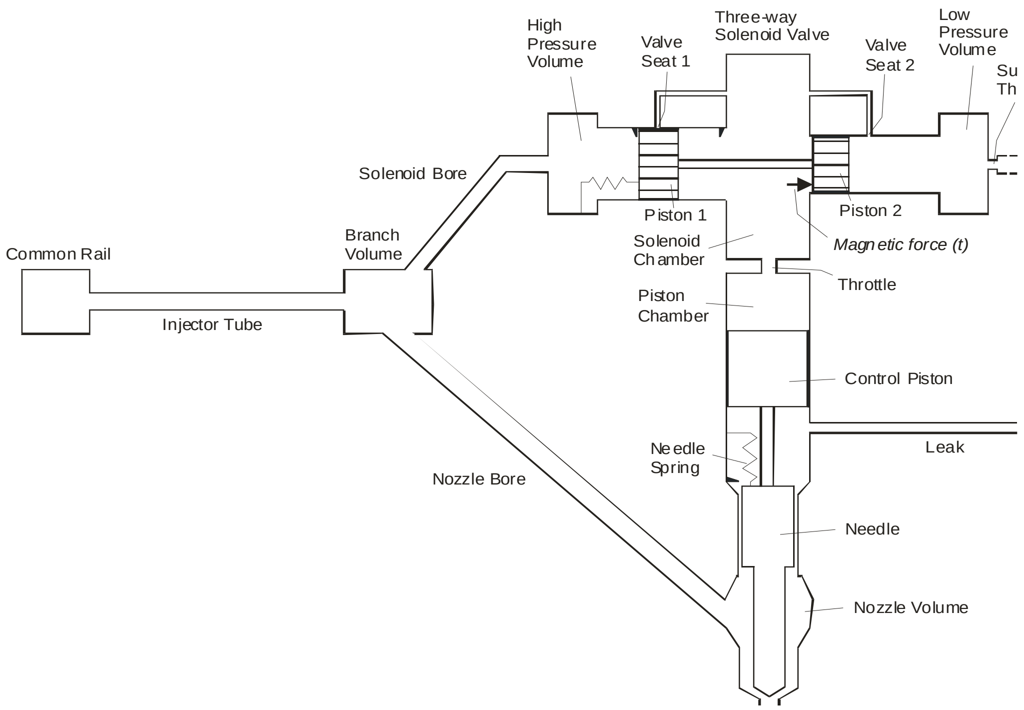


Figure 5-28: Layout of Common Rail Injector with 3/2-Way Valve (needle open)

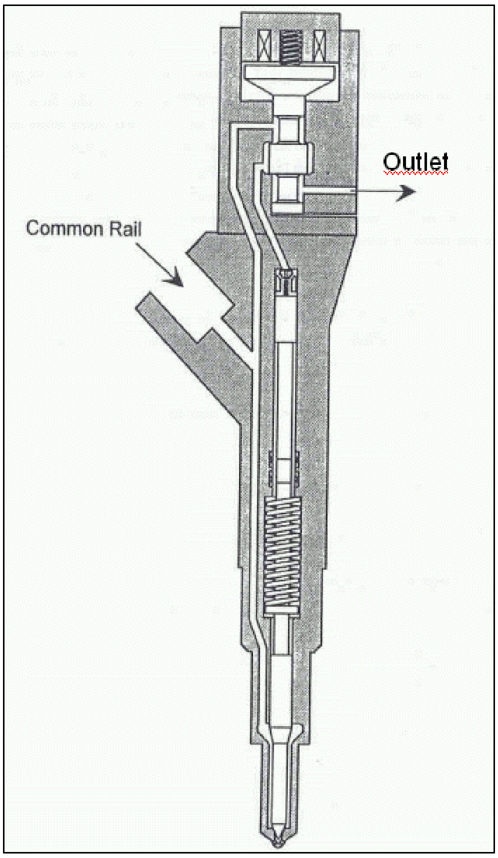


Figure 5-29: Schematic of Common Rail Injector with 3/2-Way Valve

5.2.1. Creating the Model

The BOOST Hydsim model of the injector with the 3/2-way valve is shown in *Figure 5-30*. It consists of the following elements:

- Rail Pressure 19[4] **(BOUNDARY/Pressure)**
- Injector Tube 1 **(LINE/Laplace Transform)**
- Branch Volume 2 **(VOLUME/Standard)**
- Solenoid Bore 3 **(LINE/Laplace Transform)**
- Nozzle Bore 4 **(LINE/Laplace Transform)**
- SV Volume 5 **(VOLUME/ Standard)**
- Nozzle Volume 6 **(VOLUME/Standard)**
- SV Chamber 7 **(VOLUME/Standard)**
- SV Casing 8 **(BOUNDARY/Mechanical)**
- Cylinder Pressure 20 **(BOUNDARY/Pressure)**
- Spill Volume 9 **(VOLUME/ Standard)**
- SV Piston 23 **(PISTON/Standard)**
- SV Seat 1 – 11 **(THROTTLE/Lift-controlled)**
- SV Armature 10 **[SOLENOID/ Armature(Basic model)]**
- SV Seat 2 – 12 **(THROTTLE/Lift-controlled)**
- Throttle 13 **(ORIFICE/General)**
- Piston Chamber 14 **(VOLUME/ Standard)**
- Control Piston 15 **(PISTON/Standard)**
- Needle 22 **[NEEDLE/Standard(up-to date model)]**
- Nozzle Orifice 16 **[NOZZLE/SAC (Basic model)]**
- Nozzle Holder 17 **(BOUNDARY/Mechanical)**
- Sump Throttle 18 **(ORIFICE/General)**
- Leak Pressure/Fuel Tank 21 **(BOUNDARY/Pressure)**

[4] Numbers given for elements may differ from those in your model. Reference *Figure 5-30* for numbered items.

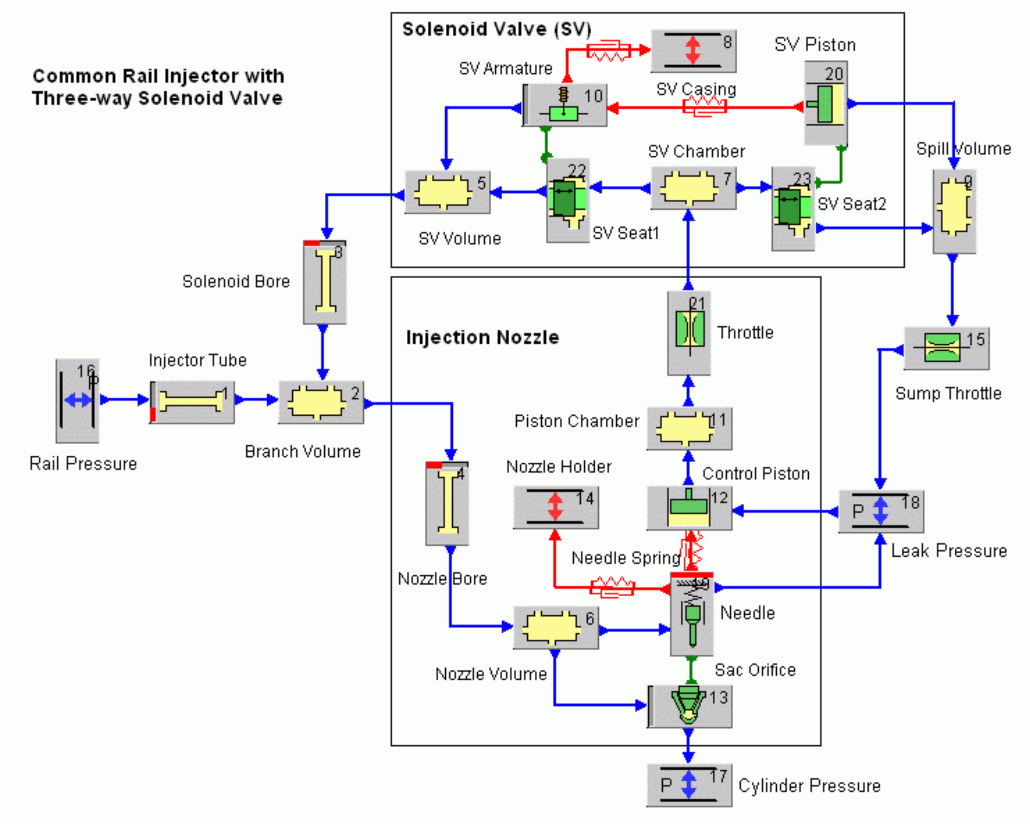


Figure 5-30: BOOST Hydsim Model of Common Rail Injector with a Three-way Valve

Refer to previous examples for information to create the system. The following is a general overview of the connections and attributes.

To make the model simpler, leakage through the needle guide and the control piston is not included here. The user is free to add leakage elements wherever he/she thinks appropriate.

The common rail pressure is chosen at 1000 bar. Injector Tube 1 connects Branch Volume 2 at the injector inlet to the rail. From this volume the system is split into two passages: one going through Nozzle Bore 4 to Nozzle Volume 6, and the other via Solenoid Bore 3 to SV Volume 5 of the Solenoid Valve.

Nozzle Volume 6 is connected to Needle 22 and Nozzle SAC Orifice 16. SAC 16 is linked to Cylinder Pressure 20, which is a Hydraulic Boundary with constant pressure of 55 bar (pressure can also be variable, of course). With the help of a special connection (green line), SAC Orifice 16 is attached to the Needle 22. Needle 22 is connected via soft spring to the Nozzle Holder 17 and, via a stiff rod, to Control Piston 15, which can move in Piston Chamber 14.

Control Piston 15 and Needle 22 have outlet connections to Leak Pressure 21, which has a hydraulic connection to the Fuel tank. It is modeled as a Hydraulic Boundary with a pressure of 1 bar.

Piston Chamber 14 is connected through Throttle 13 to SV Chamber 7. SV Chamber 7 has overall four hydraulic connections: two to Valve Pistons 10 and 23 and two to Valve Seats 11 and 12, respectively.

SV Piston 23 is attached via a special connection to Seat 12, as is SV Armature 10 to Seat 11. Furthermore, Piston and Armature are connected by a stiff mechanical joint (red line). In fact, they move practically together. However, they cannot be modeled as one piston because of two throttles which cross-sectional areas depend on the position of the pistons. These variable flow areas are represented by the two Valve Seats 11 and 12, and will be discussed later in this section.

Additionally, SV Armature 10 is connected to the SV Casing 8 via mechanical connections. This connection is a physical spring with preload, stiffness, and damping. The connection represents the velocity-dependent (viscous) damping of the piston.

SV Armature 10 and SV Seat 1 (11) have hydraulic connections to the high pressure Inlet Volume 5, which is connected via Solenoid Bore 3 to Branch Volume 2. SV Piston 23 and SV Seat 2 (12) have hydraulic outlets to the low pressure Spill Volume 9. This volume is connected via a Sump Throttle 18 to the Leak Pressure 21 (Fuel Tank).

5.2.2. Input Data and Operation

The common rail injector operates as follows. At starting position (refer to *Figure 5-27*) SV Seat 1 (11) is fully open and SV Seat 2 (12) is closed. Pressure in the SV Volume 5, Piston Chamber 14 and Nozzle Volume 6 is equal to the rail pressure 1000 bar. Needle 22 is closed because the force acting on the top of Control Piston 15 is greater than the hydraulic force under the needle guide (due to the area difference and spring preload). At a specific time instant the magnetic force is applied which moves SV Armature 10 and SV Piston 23 to the left, closing SV Seat 1 (11) and opening SV Seat 2 (12). For simplicity, in this example the constant magnetic force 500 N is chosen.

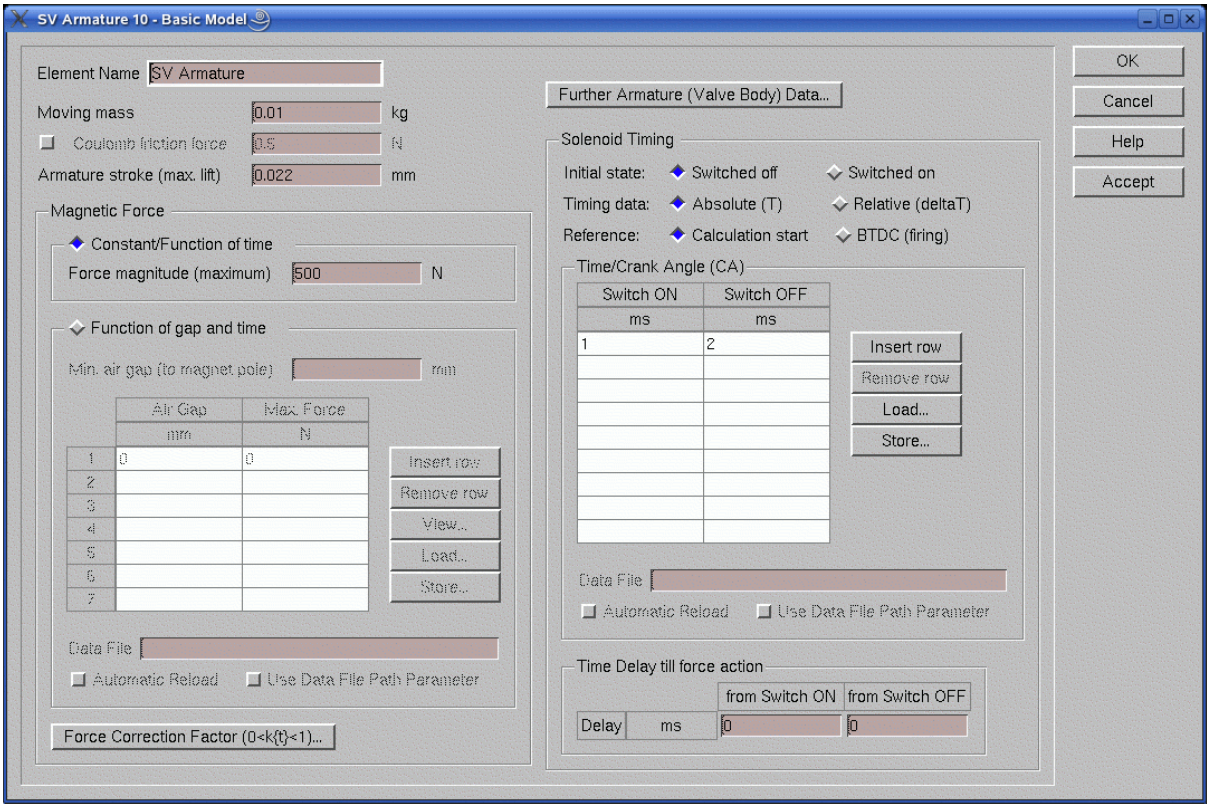


Figure 5-31: Input Dialog of Solenoid Armature (Basic model)

☞	**Note:** Within **Solenoid Armature (Basic model)** input dialog, the user can define a constant (pick-up) magnetic force independent of armature motion (coarse simplification) or gap-dependent magnetic force dependent on the actual armature position (realistic model). For a gap-dependent magnetic force, a minimum air gap and force-gap table must be specified.

In this dialog, the following data should be specified:

a. Moving mass: 10g

b. Coulomb friction force: 0,5 N (optional)

c. Armature stroke (max. lift): 0,022 mm

d. Constant force magnitude: 500 N

e. Solenoid timing (switch On/Off): 1 and 2 ms

Additional armature data has to be specified in the sub-dialog **Further Armature (Valve Body) Data**:

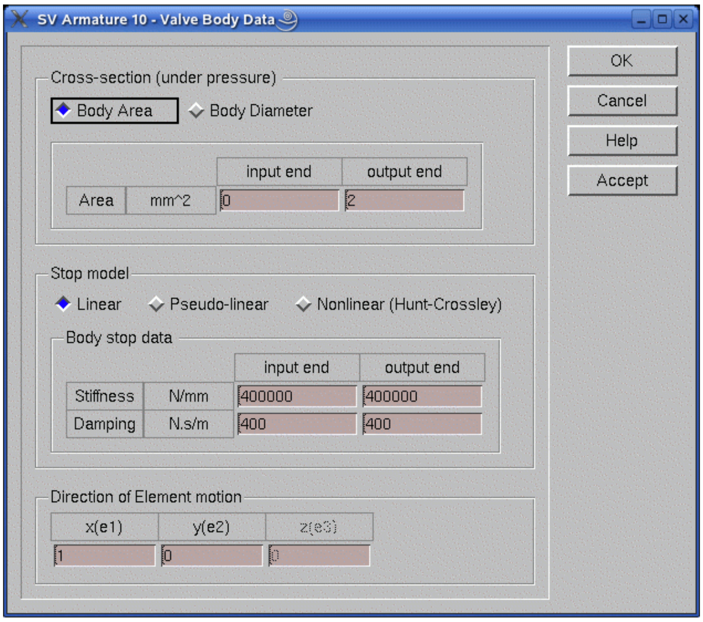


Figure 5-32: Input Dialog of Further Armature (Valve Body) Data

Specify the following parameters:

a. Body stop stiffness and damping. Thus, specify 400000 N/mm for stiffness and 400 Ns/m for damping

b. Body cross-sectional areas (under pressure). Set cross-sectional area output end to 2 mm^2. Cross-sectional area on input end is irrelevant here because there is no hydraulic connection on armature input end.

To explicitly define a variable magnetic force (as a function of time or shaft angle), the **Element | Modify** option has to be used. This is a powerful function applicable for most of the input parameters which can be varied during execution. However, it has to be used with care as the stability of integration may be affected.

As stated earlier, due to a high stiffness of the connecting rod (100000 N/mm in our case), SV Armature 10 and SV Piston 23 move practically together as one rigid body. The motion of the SV Armature 10 imposes closing of SV Seat 1 (11) and SV Piston 23 – opening of SV Seat 2 (12). The opening/closing characteristics (effective cross-sectional flow areas) are specified in the input dialogs of SV Seat 1 (11) and SV Seat 2 (12) as a function of piston displacements.

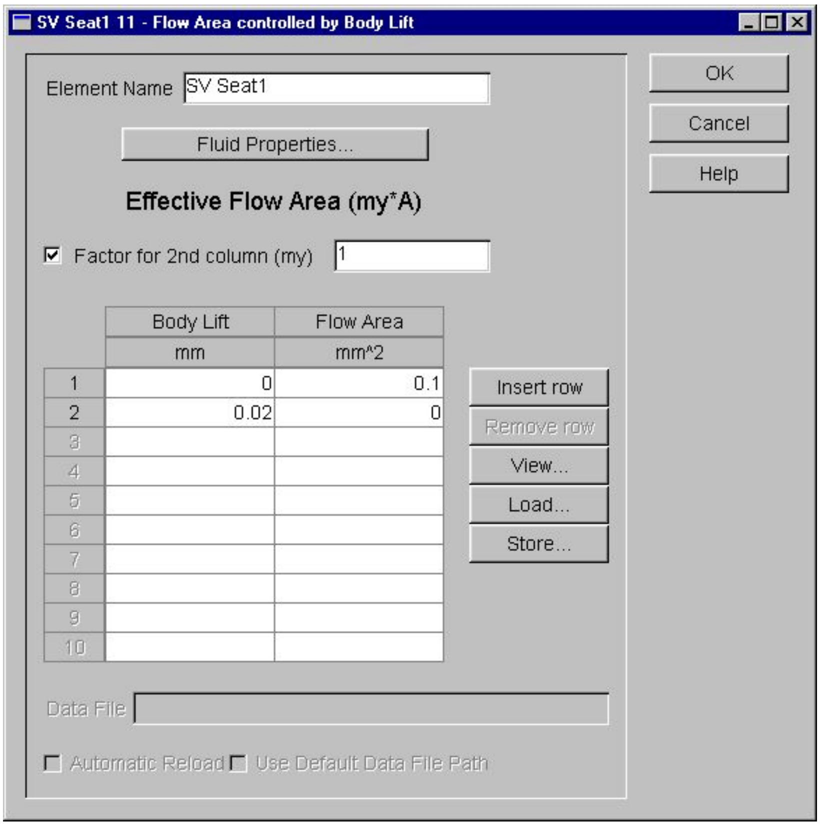


Figure 5-33: Input Dialog of Valve Seat (Lift-controlled Throttle)

Here, an effective cross-sectional flow area through SV Seat 1 (11) vs. lift of SV Armature 10 is specified. Note that at start (zero lift) the seat is fully open (effective area 0.1 mm2). In our case the closing characteristic is linear as only two values are specified. Analogously, the flow area through SV Seat 2 (12) vs. motion of SV Piston 23 is defined. This seat is closed at start. The flow areas in between these input values are linearly interpolated. The values outside the defined lift range are kept constant at the minimum/maximum bounds (no extrapolation performed).

As soon as SV Seat 2 (12) starts opening, pressure in SV Chamber 7 and Piston Chamber 14 drops down at a high rate. Fuel flows from SV Chamber 7 through SV Seat 2 (12) into low pressure Spill Volume 9 and from Piston Chamber 14 into SV Chamber 7. Pressure under the needle guide (in Nozzle Volume 6) stays nearly constant at 1000 bar. Note that the pressure under needle seat (cylinder pressure, 55 bar in our case) is very low compared to the pressure under the needle guide (rail pressure, 1000 bar), therefore the hydraulic force acting on the needle seat is negligibly small compared to the force under the needle guide. As soon as the pressure difference between the Nozzle Volume 6 and Piston Chamber 14 reaches nozzle cracking pressure (defined by the preload of the needle spring), Needle 22 lifts up and the fuel is injected into the cylinder. The injection process lasts until the actuation of the solenoid armature is switched off. The spring force (mechanical connection between SV Armature 10 and SV Casting 8) returns SV Armature 10 into its initial position (refer to *Figure 5-27*). SV Armature 10 opens the inlet SV Seat 1 (11) from the common rail side and SV Piston 23 closes the outlet SV Seat 2 (12) to the low pressure circuit. Obviously, there is a time delay between the solenoid switching and injection begin/end that can be adjusted by varying the throttle diameter, spring stiffness, volume sizes and other system parameters.

It was mentioned that a SAC type nozzle orifice is used in this model. The input data dialog of the SAC Nozzle Orifice 16 is as follows:

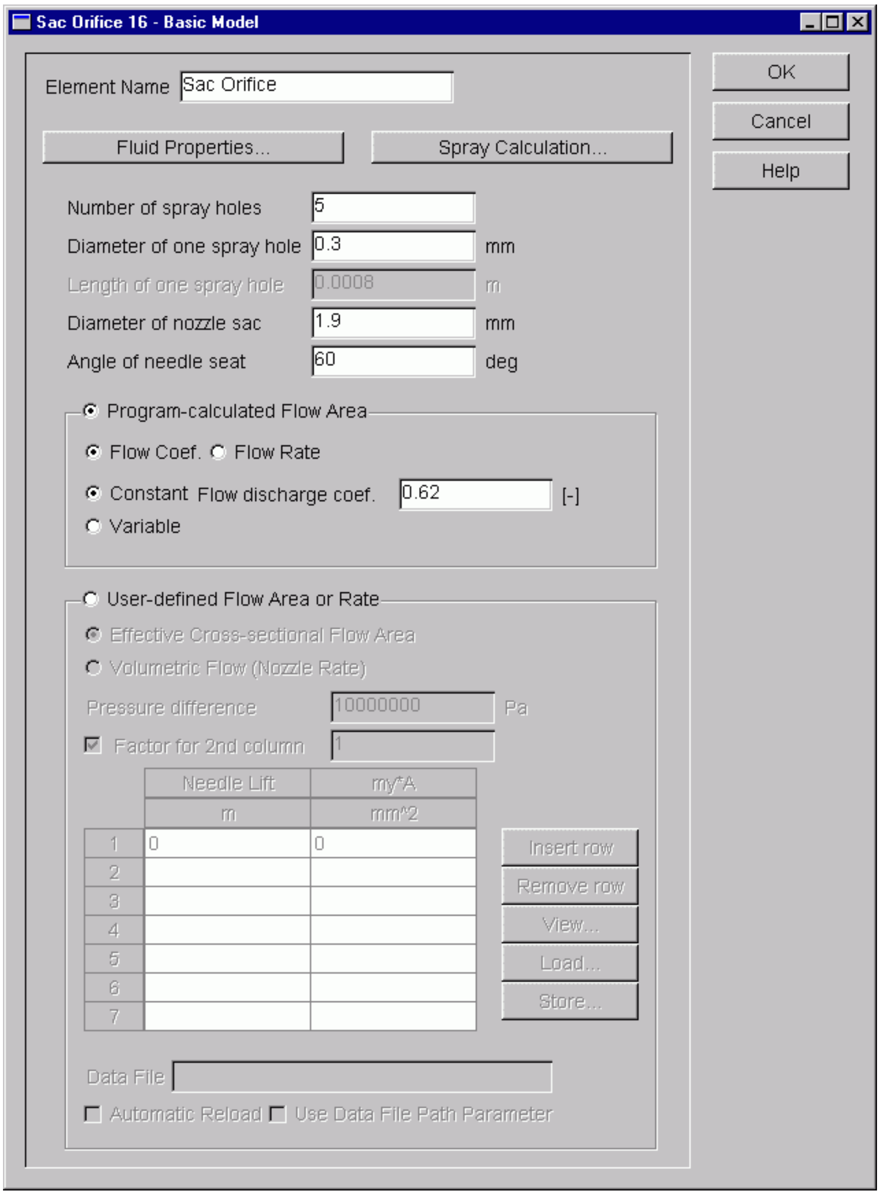


Figure 5-34: Input Dialog of Basic SAC Nozzle Orifice

The dialog of Basic SAC Nozzle Orifice is almost same as the dialog of Basic VCO Orifice (refer to *Figure 5-12*) except for one parameter – diameter of nozzle sac instead of nozzle diameter at spray holes. Here again the user has two possibilities:

- Internal BOOST Hydsim algorithm (refer to BOOST Hydsim Users Guide) can be used for calculating the effective cross-sectional flow area of the nozzle orifice. Activate **Program-calculated Flow Area** (default). This algorithm assumes that flow discharge coefficient is directly defined by the user. It may be constant or function of pressure ratio. Set it to 0.62. Next, set the angle of needle seat to 60 degrees, number of spray holes to 5 and diameter of one spray hole to 0.3 mm.

- An alternative option is to provide directly the effective cross-sectional flow area or nozzle rate vs. needle lift (e.g. calculated externally or measured). Activate **User-defined Flow Area or Rate**. The table will be activated where the user can specify the effective flow area or volumetric rate as a function of needle lift. Table data can be imported from an external ASCII file. In this type of SAC Orifice transition from hydraulic to cavitating flow is not considered, if the discharge coefficient is constant. However, for **User-defined Flow Area or Rate** option, the cavitation effects can be included in the flow area table. To check for cavitation and other effects using the BOOST Hydsim internal model, the current element has to be replaced by a more complex SAC Nozzle Orifice named SAC (Extended model). It is also accessible from **Nozzle** group. Notice that both VCO (Extended) and SAC (Extended) models account separately for the flow discharge at the needle seat and spray holes.

5.2.3. Calculation Control

This example will demonstrate the application of the **Variable-step** solver. Its typical advantage is calculation speed as the solver automatically adapts the time step according to the dynamic behavior of the system.

Before starting the calculation, specify calculation control data in **Simulation**| **Control**. Select **Variable-step** for **Solver Type** and define the **Error Type** and **Tolerance** parameters. **Calculation end time** and **Number of values to store** are the same as for the **Fixed-step** solver. **Error Type** is set to **Mixed** by default. This tolerance type is generally recommended. The other two options (**Absolute** and **Relative**) require a profound understanding of numerical behavior for choosing the tolerance value.

The recommended tolerance for hydro-mechanical models is *10^{-4}* (see below).

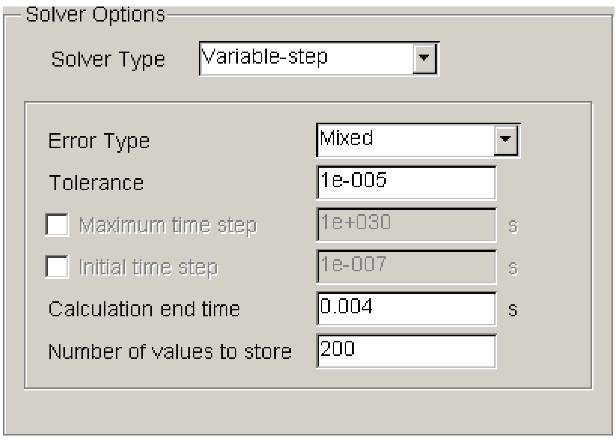


Figure 5-35: Input Dialog for Variable Step Solver

5.2.4. Running Calculation

Before starting the calculation, specify initial conditions (under **Edit**| **Initial Values** and **Element**|**Initial Values**), calculation control data (from **Simulation**| **Control**), fluid properties (from **Model**|**Fluid Properties**) and desired output data (from **Element**|**Store Results**). To be consistent with the previous example, choose `common_rail3wv` for the model name (abbreviation for common rail injector with three-way valve).

Refer to *Section 4.4* to perform the calculation.

5.2.5. Calculation Results

Refer to *Section 4.5* for loading and viewing results with IMPRESS Chart.

Example: Program-calculated Cross-Sectional Flow Area

Figure 5-36 shows some calculation results for an example with program-calculated effective cross-sectional flow area. The top graph of *Figure 5-36* depicts the **Rail pressure** (straight line, 1000 bar) and pressure variation in the **Branch Volume**, **Nozzle Volume** and two solenoid volumes: **SV Chamber** and **Spill Volume**. At the beginning, the solenoid valve is closed and pressure in these volumes stays constant at 1000 bar. At the opening of **SV Seat 2** (outlet passage) we see a sharp pressure drop in **SV Chamber**. Pressure in **Nozzle Volume** stays practically constant until the very opening of **Needle** and pressure in **Branch Volume** and **Spill Volume** begins decreasing after a small time delay, determined by the pressure wave propagation in the system. At the closing of **SV Seat 2** (outlet passage) and simultaneous opening of **SV Seat 1** (inlet passage) the pressure in **SV Chamber** steeply increases and for a short time, even exceeds the pressure in **Nozzle Volume**. The needle starts closing, and the generated shock wave causes strong pressure disturbance across the system.

At the bottom of *Figure 5-36* the input and output flow rates through **Injector Line** are shown. Due to the fuel compressibility and non-stationary frictional losses (calculated inside the line module) the output rate is substantially lower than the input rate. In both curves, the high flow oscillations can be observed. They could have a negative effect on the common rail stability (not modeled here).

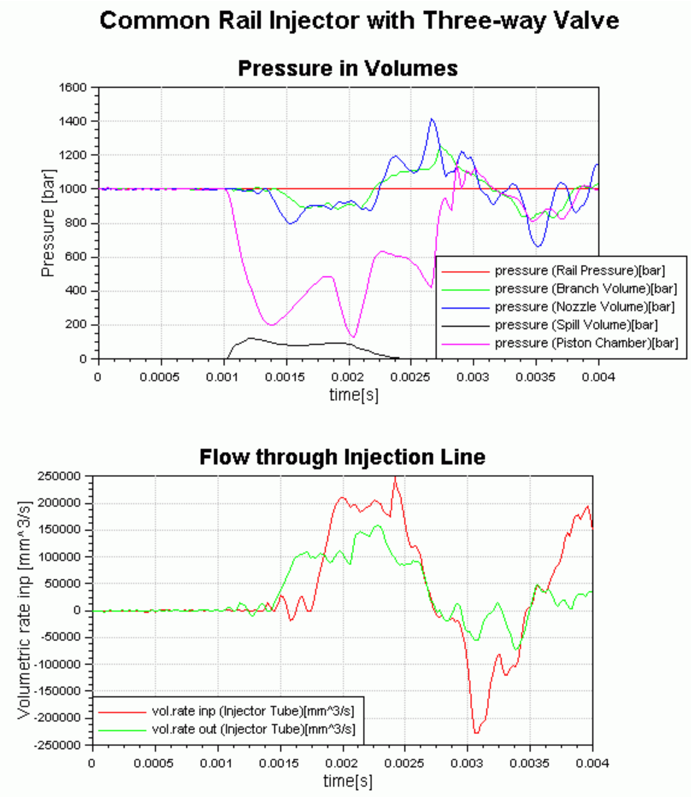


Figure 5-36: Pressure in Volumes and Flow through Injection Line

Figure 5-37 shows the motion of **Needle** and **Control Piston** (top graph), injection rate (middle graph) and effective flow area of the solenoid **Valve Seats** (bottom graph).

The user may observe a characteristic time delay between the opening of **SV Seat 2** (closing of **SV Seat 1**) and injection beginning. This time delay is controlled by the **Throttle** between **Piston Chamber** and **SV Chamber**.

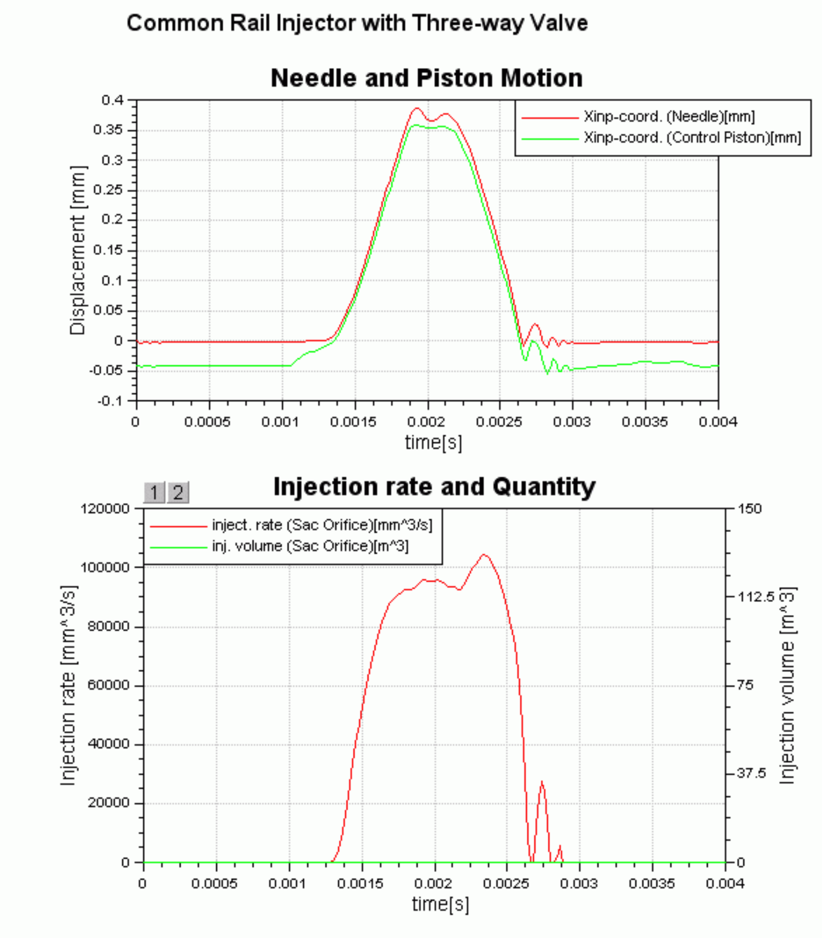


Figure 5-37: Needle and Control Piston Motion, Injection Rate and Quantity

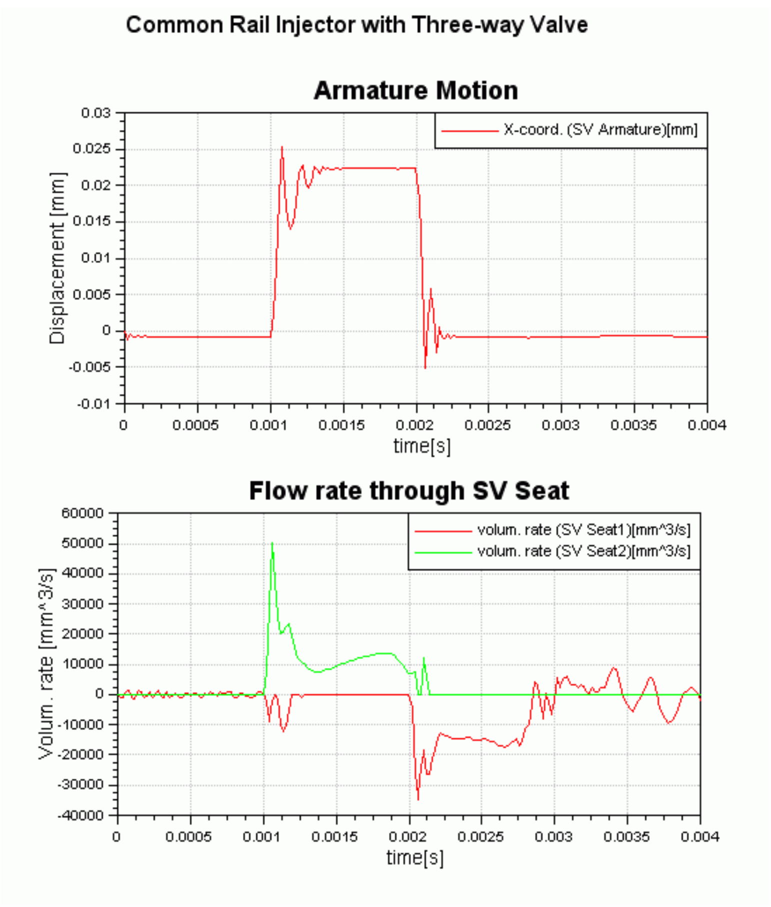


Figure 5-38: Armature Motion and Flow through Valve Seat

5.2.6. User Elements

The last group in the Element tree is **User Elements**. It allows the BOOST Hydsim model parts to be stored under a user-defined name as a new element, which can be used in the other BOOST Hydsim models. This can be done using **Edit | Modules**.

Firstly select the desired elements and their connections, which have to be stored as User element (super-element). Then open **Save as User Module** from **Edit | Modules** menu. In this example, model parts within "Injection Nozzle" and "Solenoid Valve (SV)" rectangles are used for the User Elements.

Save it under `Injection Nozzle.mod` file. The new item **Injection Nozzle** will be added in the **User Elements** group. Repeat this procedure for the "Solenoid Valve" rectangle and store it under `Solenoid Valve.mod` file. The item **Solenoid Valve** will be added in the **User Elements**.

Injection Nozzle and **Solenoid Valve** can now be used in other BOOST Hydsim models like all other elements.

Another way to add User Elements is to open **Load Module** from **Edit | Modules** menu and select the appropriate `*.mod` file.

The **Save Module** option saves selected elements under the user-defined name without adding a new item under **User Elements** group. Modules created in this way can be loaded into BOOST Hydsim model by **Load Module** option from **Edit | Modules**.

5.3. Injector with a Piezoelectric Valve

In this section, a common rail injector with a piezoelectric control valve will be described. The motion of control valve piston is controlled by the piezoelectric stack actuator. The piezoelectric stack exhibits very precise motion but its maximal displacement is usually too small (40-80 mkm). Therefore different amplifiers are used to magnify the piezo stack end motion and attain the required amplitude of control valve In this example, a general injector with a hydraulic amplifier and standard two-way servo-valve is considered. The principal scheme of the injector is shown in *Figure 5-39*.

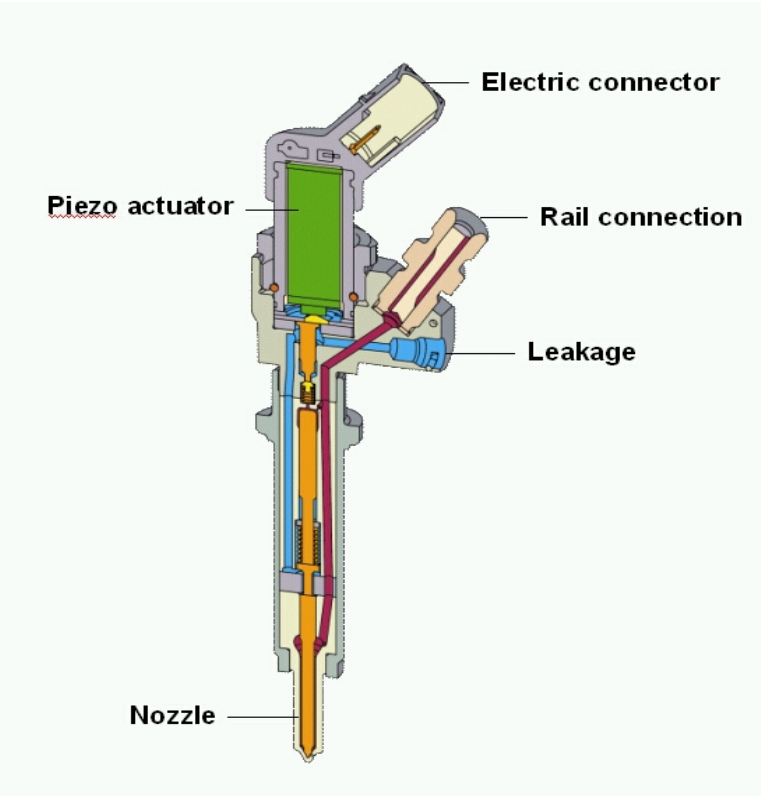


Figure 5-39: Common Rail Injector with Piezoelectric Control Valve

Schematic of piezoelectric stack actuator is shown in *Figure 5-40*. It consists of a dense pile of separately contacted layers (wafers) of electrically active ceramic material. The layers are assembled mechanically in a series and connected electrically in parallel. When input voltage is applied to stack actuator, electric field across the ceramic layers induces a mechanical strain which results in an elongation of the stack. Piezoelectric stack exhibits the rate-independent hysteresis between the electric voltage (force) and mechanical strain (displacement).

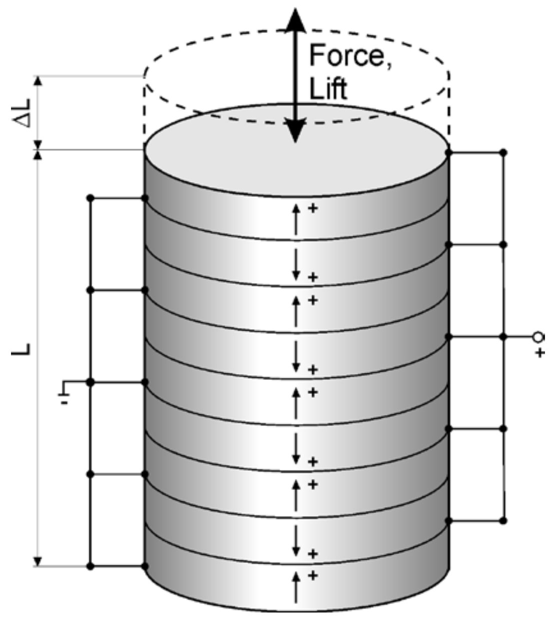


Figure 5-40: Piezoelectric Stack Actuator

Piezoelectric stack lift can be amplified by a simple device consisting of two pistons and volume between them. Motion of the second piston depends on the ratio of the cross-sectional areas (diameters) of the pistons. To magnify the piezo stack end motion, second piston must have a smaller cross-sectional area (diameter).

5.3.1. Creating the Model

To create the injector model, use the previous model of the common rail injector with two-way solenoid valve (refer to *Chapter 5.1*). In this model, remove the obsolete solenoid valve and add the piezoelectric stack actuator, lift amplifier and servo-valve. The new injector model is shown in *Figure 5-41*. Piezo actuator consists of Piezoelectric Stack 23 and Actuator Piston 14. They are connected by a very stiff mechanical connection (250000 N/mm). Hence Actuator Piston 14 practically follows the stack end motion. Hydraulic amplifier includes Volume 13 with Actuator Piston 14 on one side and Servo-valve Piston 15 on the other side. Piston 15 is connected with a small Seat Volume 6 by a hydraulic connection and with Servo-valve Seat 26 by a special connection. Thus, Piston 15 controls the opening and closing of Servo-valve.

The common rail pressure in the system is chosen at 1000 bar (Rail Pressure 18). To start a calculation from an equilibrium condition, initial pressure in Branch, Nozzle, Control and Seat Volumes has to be set to 1000 bar, too.

Compared to the injector with two-way solenoid valve in *Chapter 5.1*, the new elements are:

- Amplifier Volume 13 **(VOLUME/Standard)**
- Actuator Piston 14 **(PISTON/Standard)**
- Servo-valve Piston 15 **(PISTON/Standard)**
- Spring Support 16 **(BOUNDARY/Mechanical)**
- Piezo Actuator 23 **(PIEZO/Stack Actuator)**
- Servo-valve Seat 26[5] **(THROTTLE/Lift-controlled [Slide])**

[5] Numbers given for elements may differ from those in your model. Reference *Figure 5-41* for numbered items.

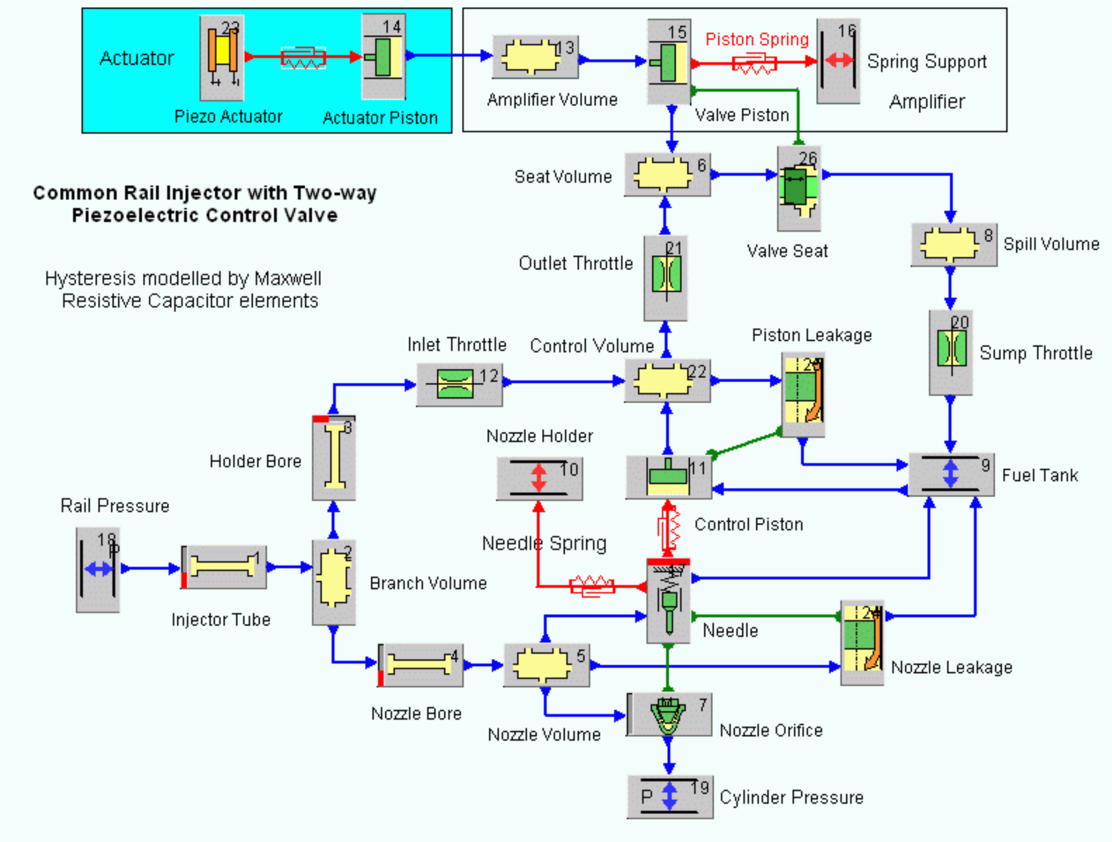


Figure 5-41: Basic BOOST Hydsim Model of Common Rail Injector with Piezoelectric Control Valve

The operation principle of the injector with piezoelectric servo-valve is the following. When input voltage is applied to stack actuator, electric field across the ceramic layers generates the motion of Piezo stack 23 and Actuator Piston 14. The lift of Actuator Piston is magnified by hydraulic amplifier and transferred to the Servo-valve Piston 15. Remember that the cross-sectional area of Valve Piston 15 is smaller than the cross-sectional area of Actuator Piston 14. The resulting displacement of Valve Piston 15 controls the seat flow area of Servo-valve Seat 26. The rest of the system operates in the same way as the solenoid-controlled injector described in *Chapter 5.1*. With an appropriate input voltage characteristic the Valve Piston can be kept at any intermediate position between input and output stops, thus allowing to achieve different shapes of the injection rate. This is not feasible with a standard solenoid-controlled injector.

5.3.2. Input Data

In this section the input data of the new elements is discussed: Piezo Actuator, Actuator Piston, Valve Piston and Valve Seat.

Initial volume of Amplifier Volume 13 is set to 80 mm^3 and initial pressure to 1.5 bar.

Piezo Actuator element allows to define the piezoelectric stack model with or without hysteresis. There are two options to define hysteresis behavior:

- Maxwell Resistive Capacitor (MRC) elements
- Normalized Stroke – Voltage diagram (idle)

The second option (Stroke – Voltage diagram) gives exact results only for the idle case. However, for moderate loads it can be used with satisfactory accuracy, too.

The Maxwell Resistive capacitor parameters (electric stiffness and break voltage) can be estimated by propagating experimental results through the model of one ceramic layer. Other parameters (mass, mechanical stiffness and damping) can be measured or determined from the properties of piezoelectric ceramic (electric capacitance and piezo gain or electromechanical transformer ratio).

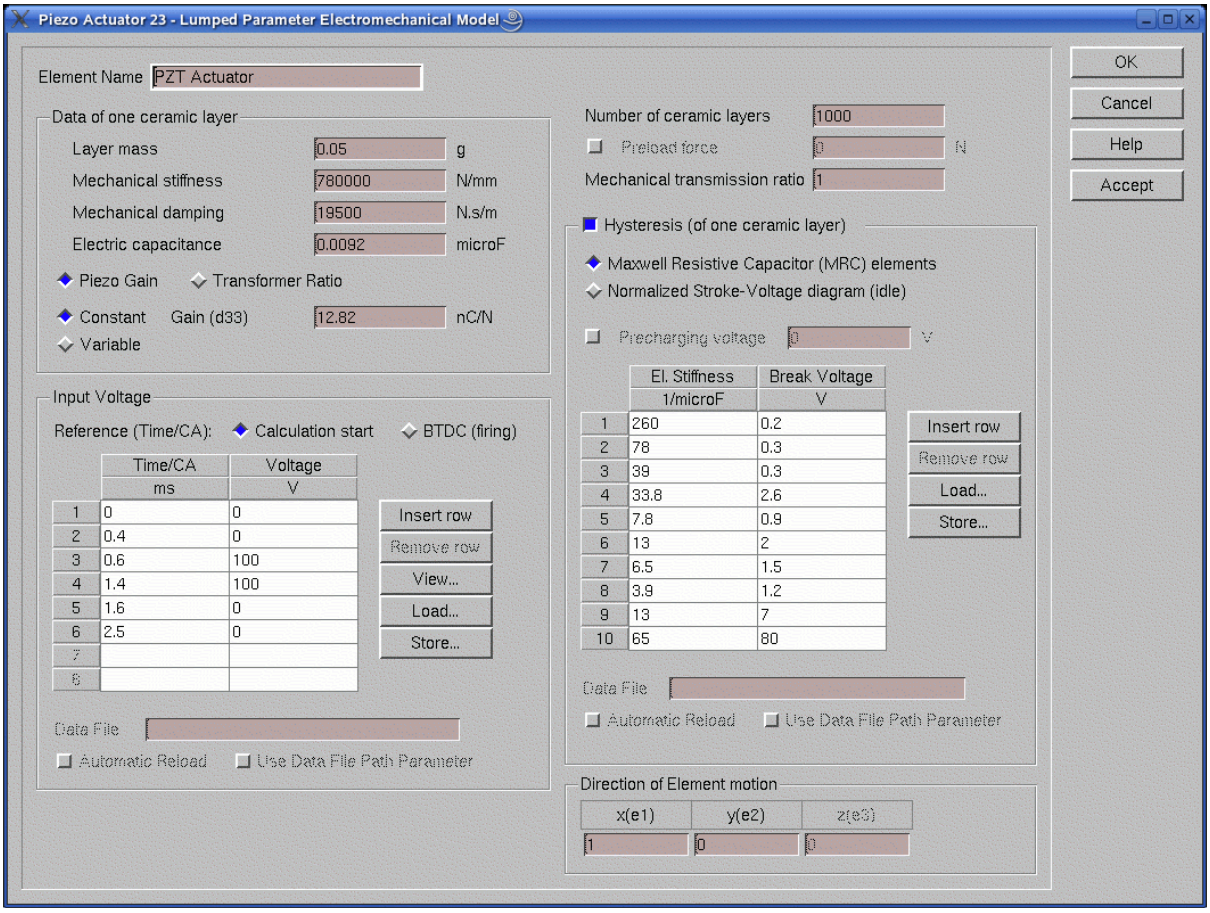


Figure 5-42: Input Dialog of Piezo Stack Actuator (MRC option)

In this example, apply input voltage pulse with amplitude of 100 V. Total number of ceramic layer is set to 130.

Note: It is formally possible to define input parameters for the whole Piezo Stack Actuator as one layer (number of ceramic layers set to 1). In this case, total mass, total mechanical stiffness, total damping and total electric capacitance of Piezo Actuator have to be specified in the input field Data of one ceramic layer. Total electric stiffness of MRC elements has to be equal to the electric stiffness of one ceramic layer divided by the number of layers.

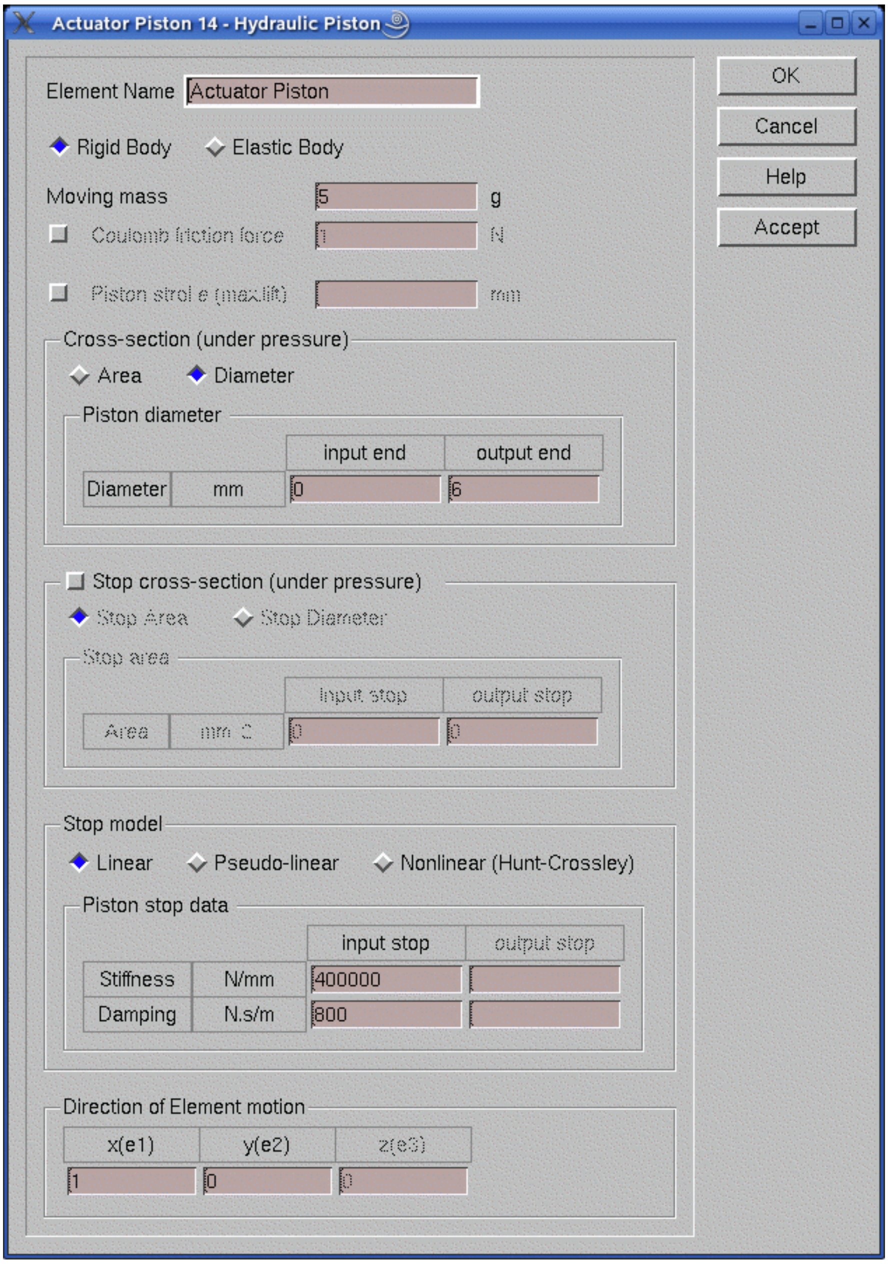


Figure 5-43: Input Dialog of Actuator Piston

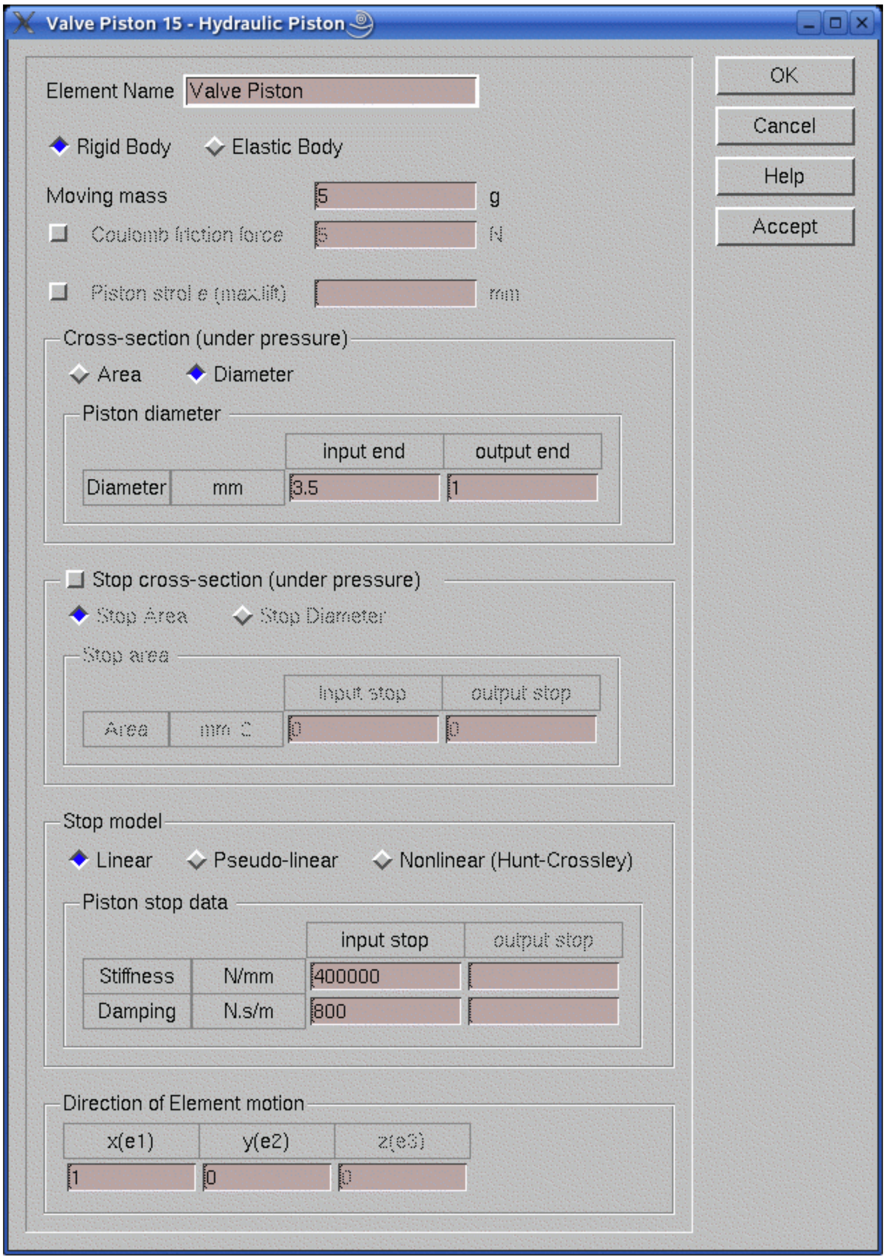


Figure 5-44: Input Dialog of Servo-valve Piston

Figure 5-43 and *Figure 5-44* show the input dialogs of Piezo Actuator Piston and Valve Piston. The user may observe that Valve Piston has a smaller diameter on input end (3.5 mm) than Actuator Piston on output end (6 mm).

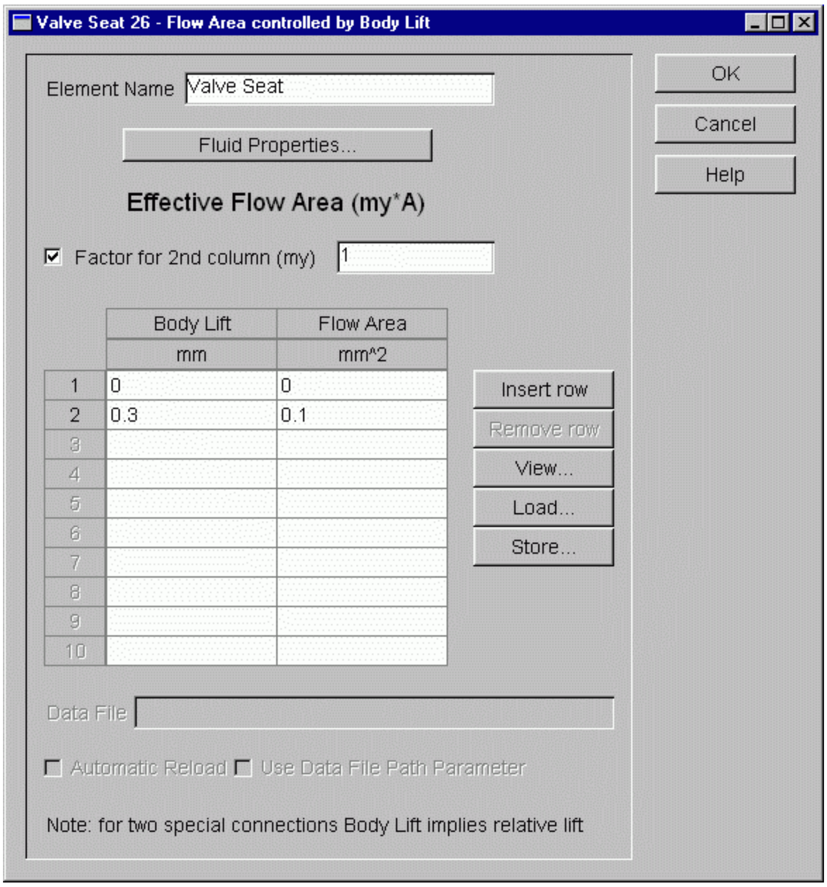


Figure 5-45: Input Dialog of Servo-valve Seat

In *Figure 5-45*, the effective cross-sectional flow area vs. Valve Piston lift is specified. Instead of the effective flow area, the user can specify the geometric flow area in the table and enter the flow coefficient as **Factor for 2nd column (μ)**. Note that at zero lift servo-valve is fully closed. Valve opening/closing characteristic is linear, thus only two rows need to be defined. The flow areas in between the rows are linearly interpolated. Flow area values outside the defined lift range are kept constant at the minimum/maximum bounds (no extrapolation performed).

Save this model under `comrail_piezo1_mrc.hyd`.

5.3.3. Running Calculation

Refer to *Section 4.4* for running the calculation.

5.3.4. Calculation Results

Refer to *Section 4.5* for loading calculation results in the Impress Chart window.

Typical calculation results are shown in *Figure 5-46* and *Figure 5-47*. Piezoelectric stack end and Actuator Piston lifts are practically the same and reach almost 100 microns (in practical applications it is lower). Actuator Piston motion is very sensitive to the pressure in Amplifier Volume (up to 200 bar) which depends on external load from the injector. Servo-valve Piston amplitude reaches 230 microns, i.e. Actuator Piston Lift is amplified 2.3 times. Time delay between the energizing of Piezo Actuator and opening of Needle is about 0.8 ms.

The shape of Needle motion, injection rate and flow through Inlet/Outlet Throttles is similar to the corresponding results of the common rail injector with two-way valve (refer to *Figure 5-17* and *Figure 5-18*). Their analysis is an exercise for the user.

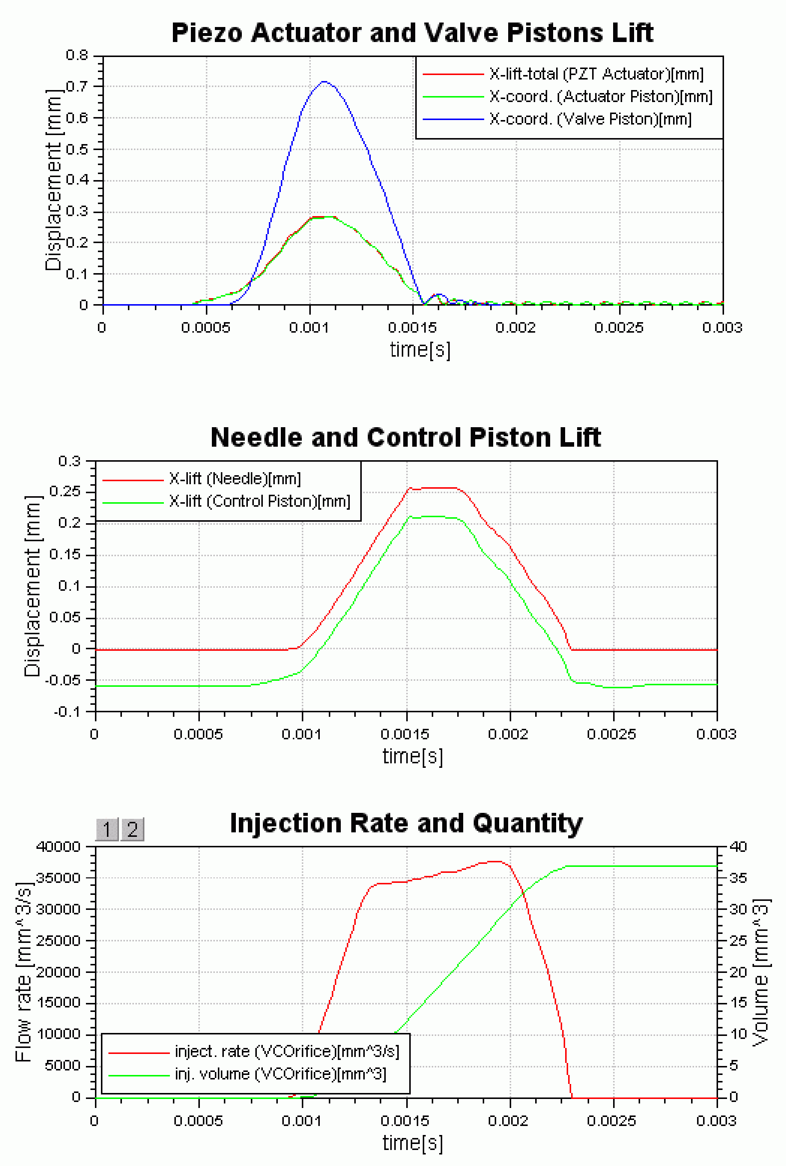


Figure 5-46: Motion of Piezo Actuator, Needle and Diverse Pistons and Injection Parameters

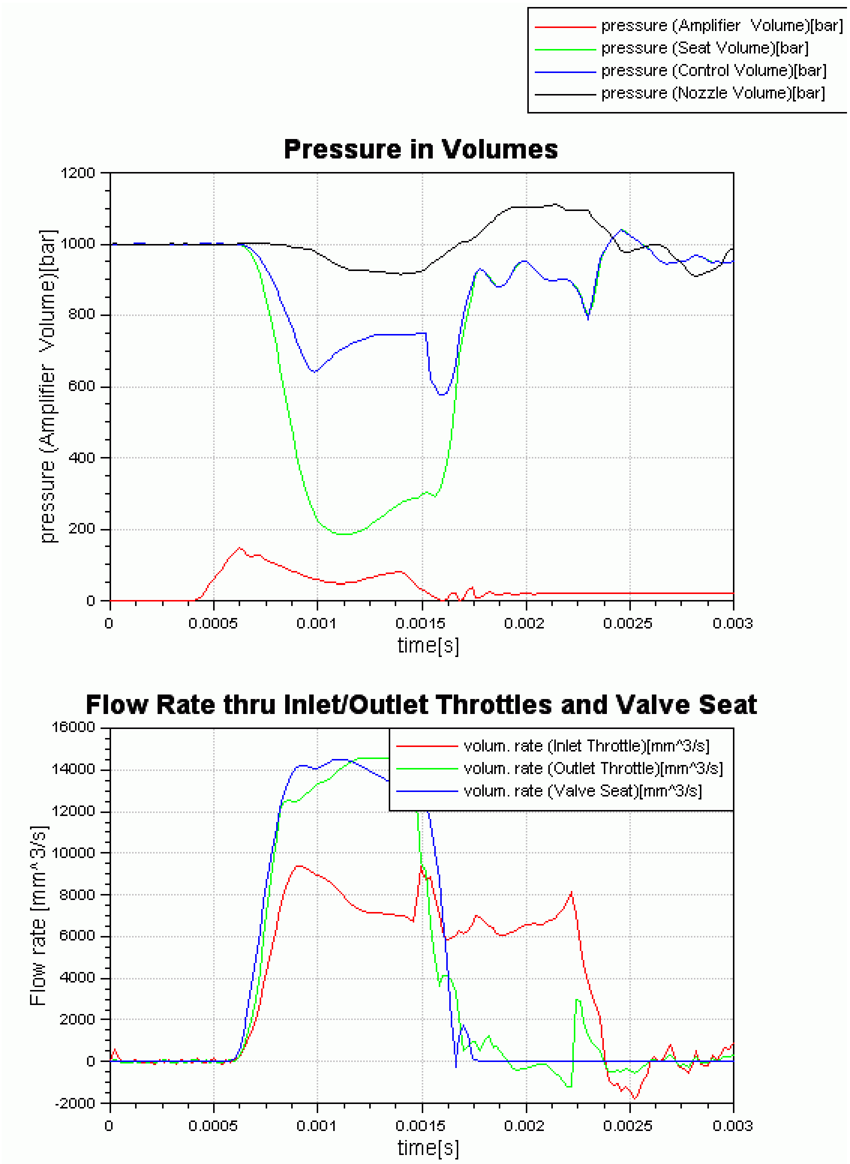


Figure 5-47: Pressure in Diverse Volumes and Flow Rates through Throttles and Servo-valve Seat

5.3.5. Model with Stroke – Voltage Diagram

In the previous example, hysteresis of piezoelectric stack is represented by Maxwell Resistive Capacitor (MRC) elements. Determining Maxwell parameters is not at all simple due to propagating experimental results. Hence another option of defining hysteresis behavior is available: **Normalized Stroke – Voltage diagram (idle)**. Although this option is primarily appropriate for idle case, i.e. the absence of any external mechanical load, it may be used with some adjustments for the load case, too.

Normalized Stroke – Voltage diagram) is defined for idle case and it may be either specified by Piezo manufacturer or determined by measurements according to the following equation:

$$X_{norm} = \frac{X}{X_{max}} \quad \text{,where:}$$

X_{norm} .... normalized stroke (-)

X .... idle stroke for applied voltage(m)

X_{max} .... maximum idle stroke (m)

Open the input dialog of Piezo Actuator and activate `Normalized Stroke – Voltage diagram (idle)` for defining hysteresis behavior of one ceramic layer.

Define the Normalized Stroke-Voltage table according to the data in *Figure 5-49* and set maximum idle stroke of one layer on 9 microns. Here a correction of maximum idle stroke is made from 9.18 microns, which is a correct value, to 9 microns in order to get more accurate Piezo displacement in load case.

Additionally, switch on `Piezo Gain (d33)`) and specify constant piezo gain parameter on 166.7 nC/N. This value results from the following relationship:

$$d_{33} = \frac{n}{k_1}$$, where:

d_{33} .... piezo gain (C/N)

n .... electromech. transformer ratio (10 C/m)

k_1 .... mech. stiffness of one ceramic layer (6E+7 N/m)

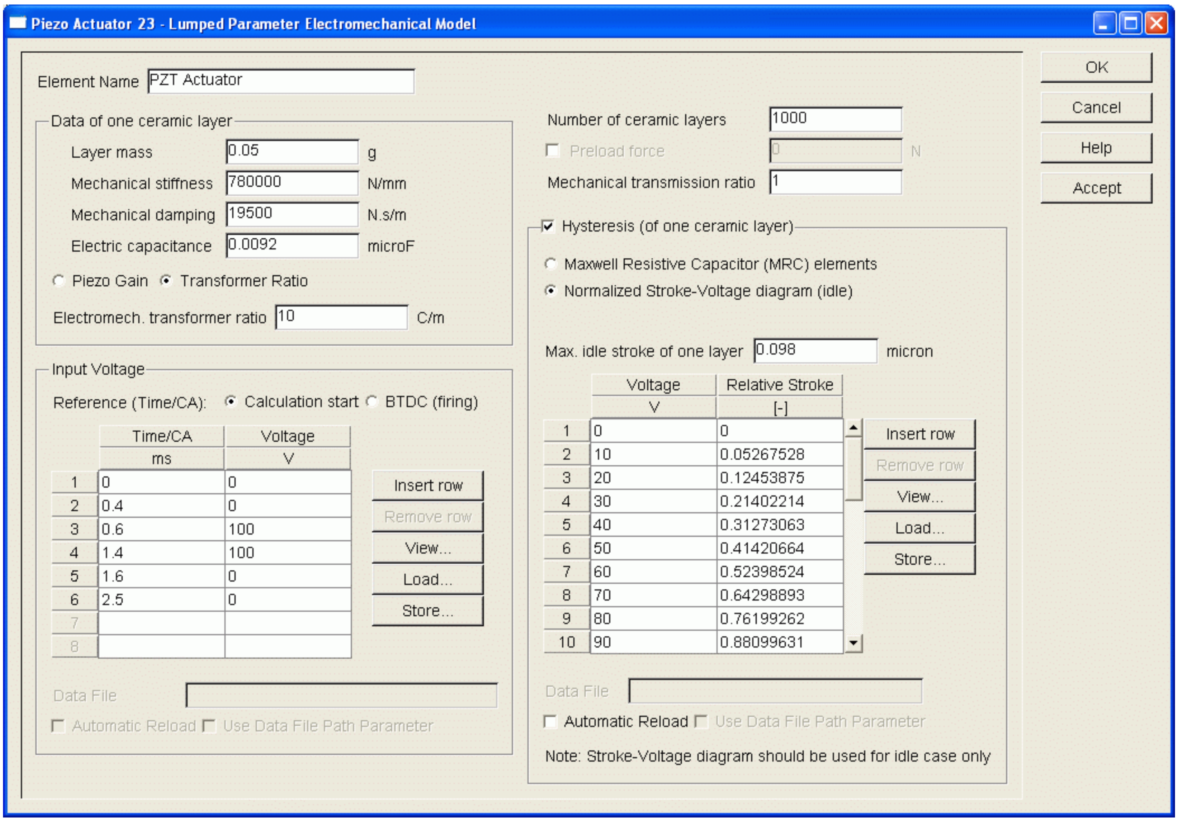


Figure 5-48: Input Dialog of Piezo Actuator (SVD option)

Note: Input voltage must be specified within range of voltage in Normalized Stroke- Voltage diagram, i.e. within its upper and lower limits. Otherwise, BOOST Hydsim will issue an error message.

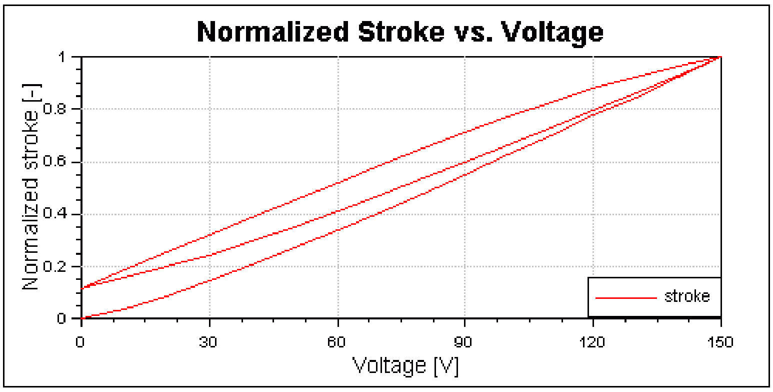


Figure 5-49: Normalized Stroke – Voltage diagram

Save this model under a different name, for instance `comrail_piezo1_svd.hyd` and run the calculation.

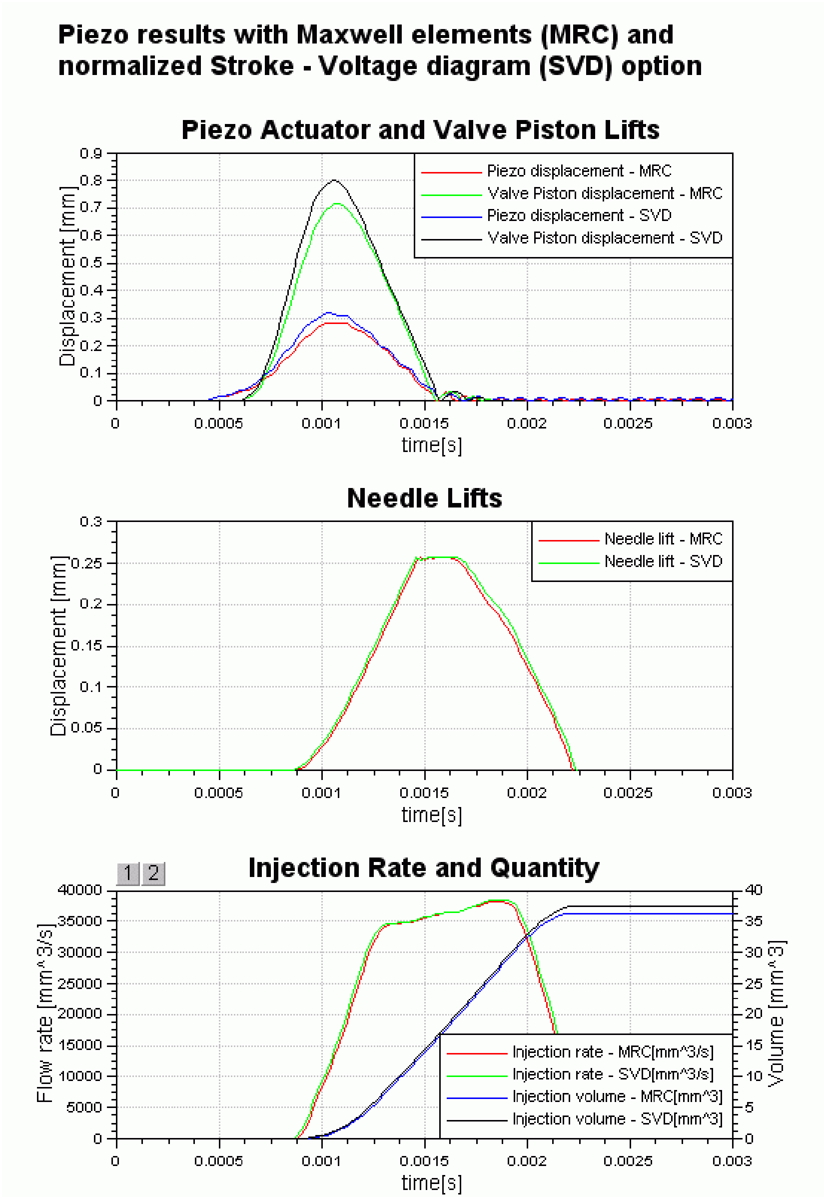


Figure 5-50: Result Comparison for MRC and SVD Options of Piezo Stack Modeling

Figure 5-50 shows compared results with both options for defining Piezo Stack Actuator hysteresis behavior – Maxwell Resistive Capacitor (MRC) and Normalized Stroke –Voltage diagram (SVD) option. The results are not the same because the Normalized Stroke – Voltage diagram is not accurate for the load case, but it can be adjusted by decreasing of maximum idle stroke parameter.

5.4. Model of CRI3 Piezoelectric Injector

In this section the model of a commercial piezo-controlled common rail diesel injector (CRI3) will be described. This injector belongs to one of the latest generations of the piezoelectric injectors. The injector model corresponds to the real design. However, the input data is modified and can be used only for education purposes.

CRI3 piezo injector operates much faster than the traditional solenoid-controlled injector. Therefore it can produce more injections per engine cycle. It can generate multiple pulses (pilot, pre-, main and post injection) and manage variable lift of the injector needle. In combination with modern combustion processes, this allows to enhance the engine performance, improve emission quality and reduce noise.

The main new features of the CRI3.0 injector are sharp injector rate, small pilot injection (PI) quantity (0.5 mm3), no PI-plateau, reduced tolerances for EU IV emission standard, multiple injections (up to 5 per cycle), reduced static and dynamic forces (wear), no leakage, fail-safe-principle, low noise, compact design and upgrade of rail pressure to 180 MPa (CRI3.1) and 200 MPa (CRI3.2), respectively.

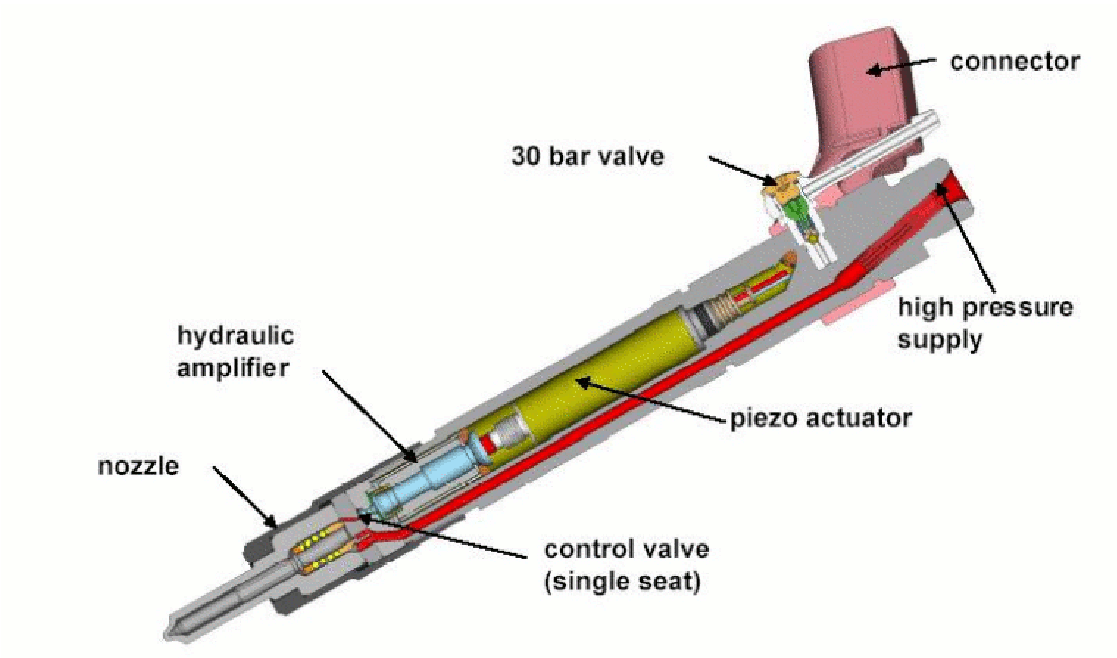


Figure 5-51: Schematic of CRI3 Piezoelectric Injector

The main assembly components of the CRI3 injector are the piezo actuator, hydraulic amplifier (coupler), control valve and nozzle. The injection process is regulated via the energizing duration of the piezo actuator which governs the motion of the control valve. Function of the control valve is shown in Figure 5-52. At the starting position no voltage is applied on the actuator.

Hence the control valve seat is closed and the high pressure area (up to 160 MPa) is separated from the low pressure (1 MPa) as shown in Figure 5-52, 1. Pressure balance across the needle keeps the nozzle closed. Activation of the piezo actuator opens the control valve and at the same time closes the bypass throttle (refer to Figure 5-52, 2). Pressure in the control volume continuously drops down and the needle starts opening. Needle opening velocity depends on the area ratio between the inlet and outlet throttle.

At maximum rail pressure 160 MPa this velocity is about 1 m/s. The closing of the valve results in the fast refilling of the control volume through the inlet and bypass throttles. The outlet throttle becomes in this case an inlet throttle as shown in Figure 5-52, 3. Pressure in the control volume increases and the needle starts closing.

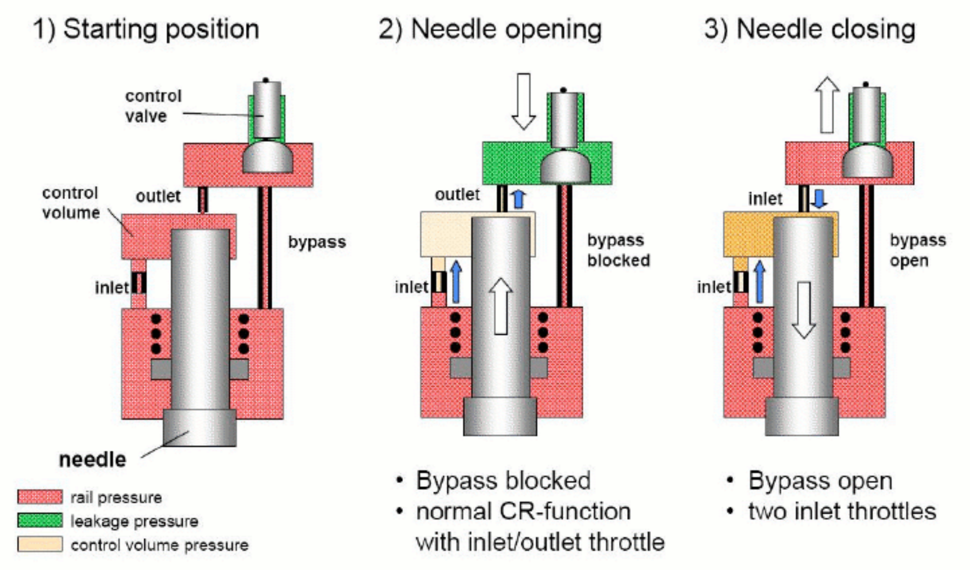


Figure 5-52: Function of Control Valve with Inlet, Outlet and Bypass Throttles

The piezoelectric stack actuator is the heart of the system model. It consists of a dense pile of separately contacted layers (wafers) of electrically active ceramic material. The layers are assembled mechanically in a series and connected electrically in parallel. When input voltage is applied to the stack actuator, electric field across the ceramic layers induces a mechanical strain, which results in an elongation of the stack. Piezoelectric stack exhibits the rate-independent hysteresis between the electric voltage (force) and mechanical strain (displacement).

The piezoelectric stack exhibits a very precise motion but its maximal (idle) displacement is usually too small (40-50 μm). Amplifiers are used to magnify the piezoelectric stack end motion and attain the required amplitude of the control valve. Hydraulic amplifier of our injector (refer to Figure 5-53) consists of the two pistons and a volume between them. Second (right) piston is connected to the control valve body. Its motion depends on the ratio of the cross-sectional areas of the pistons. To magnify the piezo stack end motion, the second piston has a smaller diameter than the first one.

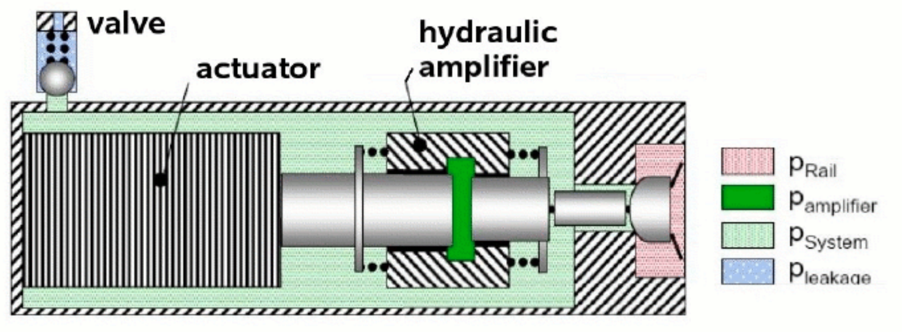


Figure 5-53: Piezo Actuator, Hydraulic Amplifier and Control Valve

5.4.1. Creating the Model

The model consists of the fuel rail with the supply pipe, injector and Bosch tube. It includes the detailed model of the nozzle (microsac) with elastic needle, throttle plate with inlet, outlet and bypass orifices, control valve and high-pressure connection between the rail and the injector and rail and throttle plate. In the model the full dynamic model of the control valve with valve body is created. Additionally, the model contains the hydraulic coupler and piezoelectric actuator.

The operational principle of the control valve with inlet, outlet and bypass throttles is shown in Figure 5-52. Note that maximum lift of control valve body is 0.035 mm while maximum needle lift is 0.9 mm. At normal operation this lift should not be reached, i.e. the needle always exhibits ballistic motion. At the closing of the control valve the outlet throttle acts like an additional inlet throttle, thus accelerating the closing of the needle.

The schematic of the hydraulic coupler (amplifier) and piezoelectric actuator is shown in Figure 5-53. Hydraulic coupler consists of the two pistons of different diameters connected by a volume. The upper (larger) piston is directly attached to the piezoelectric stack end. The lower (smaller) piston is connected to the control valve body. Under different load conditions the piezoelectric stack lift varies from 25 to 32 μm. The transmission ratio of the hydraulic coupler is about 1.4, so the lift of the control valve body is 35 μm. The valve body lift cannot get higher because it is limited by the throttle plate. The actuation voltage of the piezoelectric stack stays constant at 110 V till 60 MPa rail pressure. From this point it linearly increases to 160 V at 160 MPa rail pressure. The system pressure in the volume surrounding the hydraulic coupler and piezo actuator is kept at 1 MPa. This allows refilling the coupler volume between the injection events.

One of the specific features of the CRI3 piezo injector is the new design of the injection needle and nozzle. The needle guide does not have a circular shape (as for standard injectors) but contains four longitudinal cuts (segments) along which the fuel from the spring chamber is delivered into the nozzle chamber. In this way the needle is better balanced and the nozzle housing does not have any bore for the fuel delivery. Note that at different rail pressures the flow pattern though needle guide segments is somewhat different, causing the additional throttling up to 7 MPa at high rail pressure.

For the modeling of this-type of nozzle geometry and flow, a new Nozzle element – Flow along Non-circular Needle Guide Throttle - is developed in BOOST Hydsim v4.5. The schematic of the nozzle needle with the guide cross-section is shown in Figure 5-54.

The input dialog of the new element is shown in Figure 5-55. The complete BOOST Hydsim model of the injector is depicted in Figure 5-56.

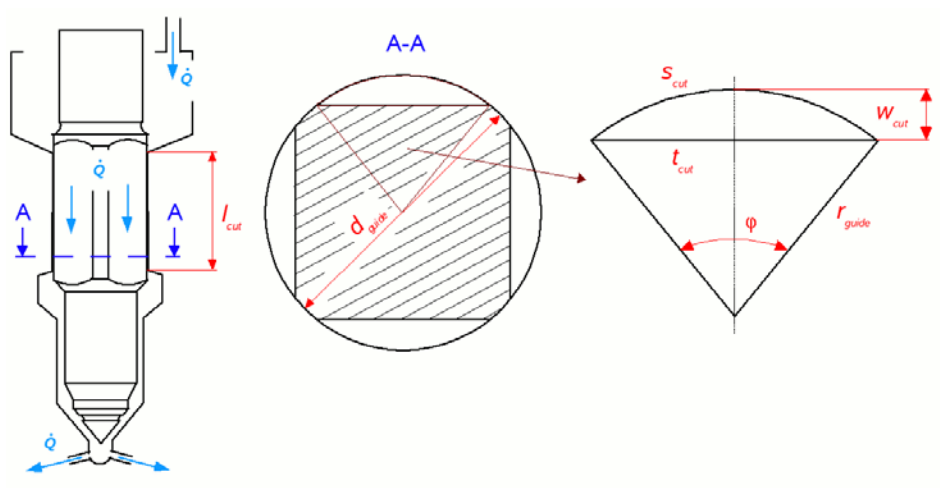


Figure 5-54: Schematic of CRI3 Nozzle Needle with Non-circular Guide

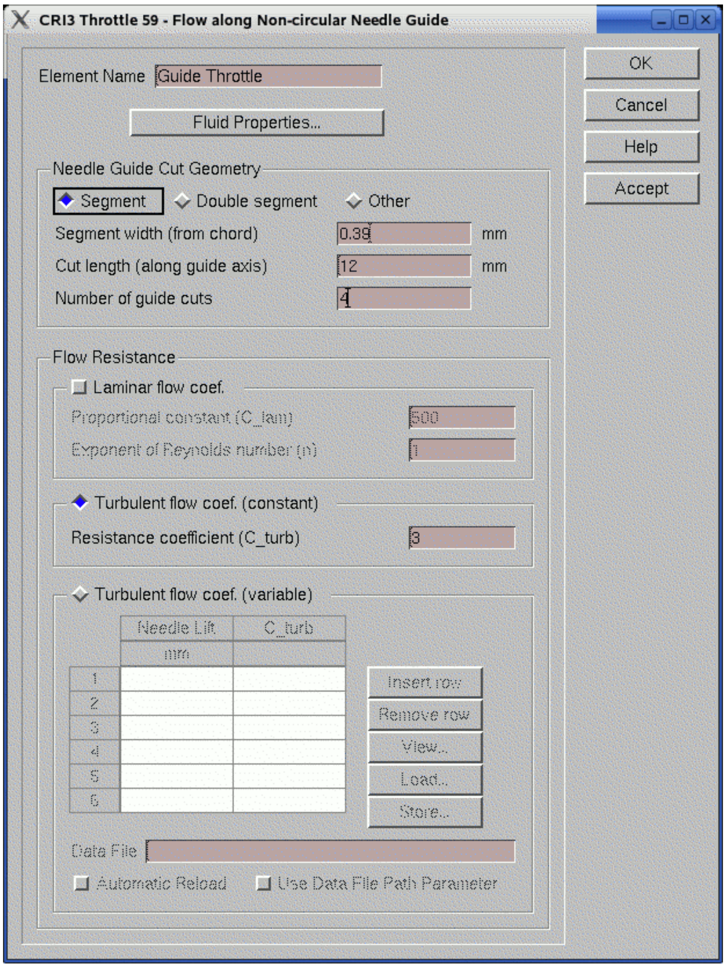


Figure 5-55: Input Dialog of Non-circular Guide Throttle

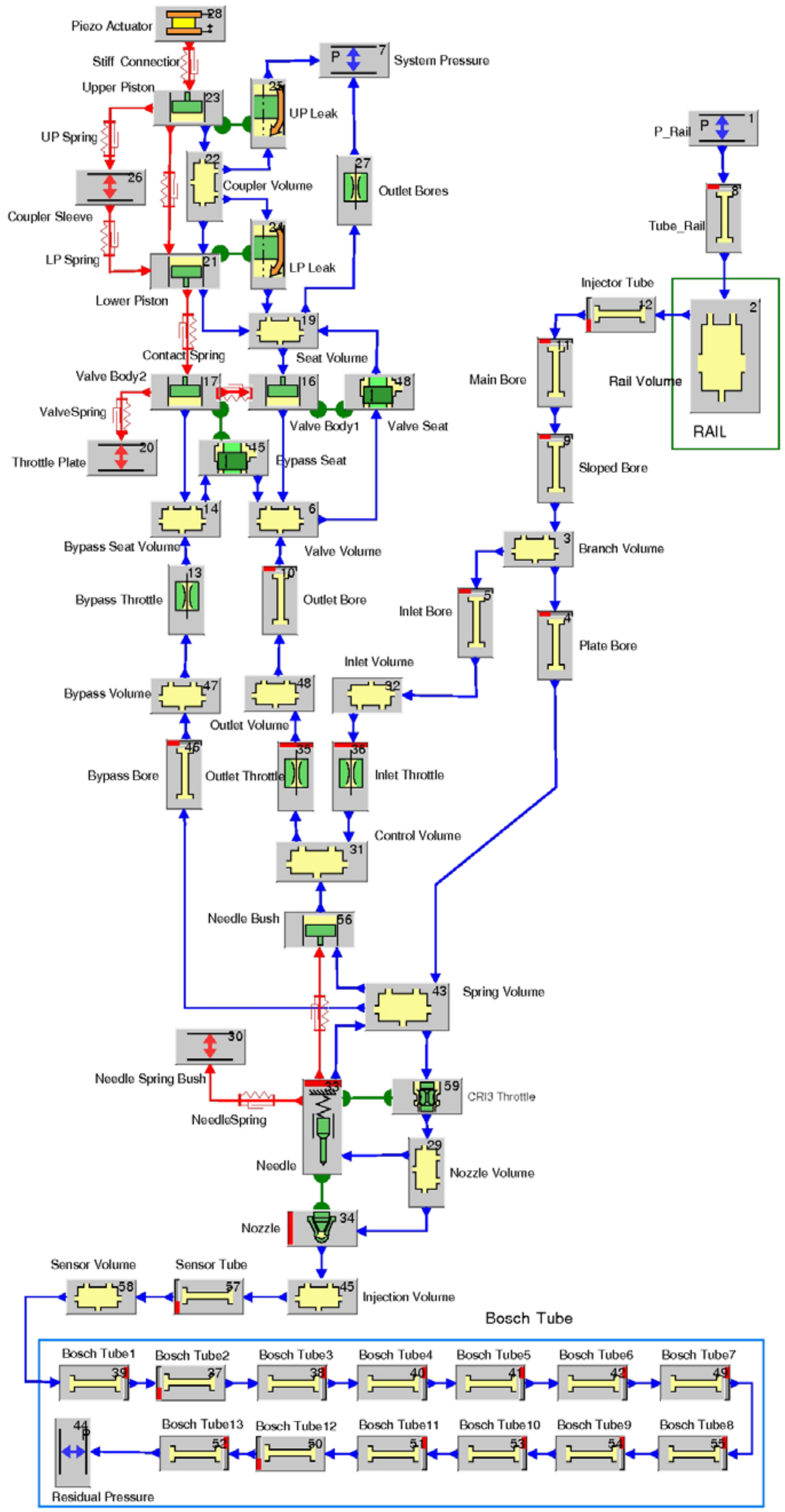


Figure 5-56: BOOST Hydsim Model of CRI3 Injector with Bosch Tube

5.4.2. Running Calculation

Refer to *Section 4.4* for running the calculation.

5.4.3. Calculation Results

Refer to *Section 4.5* for loading calculation results in the IMPRESS Chart window.

Selected calculation results for two load cases are shown in Figure 5-57 and Figure 5-58. For convenience, the results are plotted in the crank angle domain. Case 1 (Prail_780) is a full load condition with the medium rail pressure of 78 MPa. Case 2 (Prail_35) is a part load condition with the low rail pressure of 35 MPa. Both cases consist of 2 pilot and 1 main injection. The upper graphs in Figure 5-57 depict the pressure traces in the nozzle, control and sac volumes. Depending on the injection conditions, the pressure oscillations in certain volumes exceed the nominal rail pressure 15 to 20%. Middle graphs show the needle lift on input and output side. The needle is modeled as an elastic body, therefore the input and output coordinates are not same. Note that maximum needle lift is 0.9 mm (900 μm). However, at real injector operation it should never be reached. Bottom graphs show the comparison between the calculated and measured pressure in Bosch tube. In case of linear acoustics, this pressure is directly proportional to the injection rate. However, for variable fuel properties (pressure-dependent bulk modulus and density) non-linear sound waves are likely to propagate within the system. Therefore direct comparison of the pressure in Bosch tube is preferable for the accurate model validation.

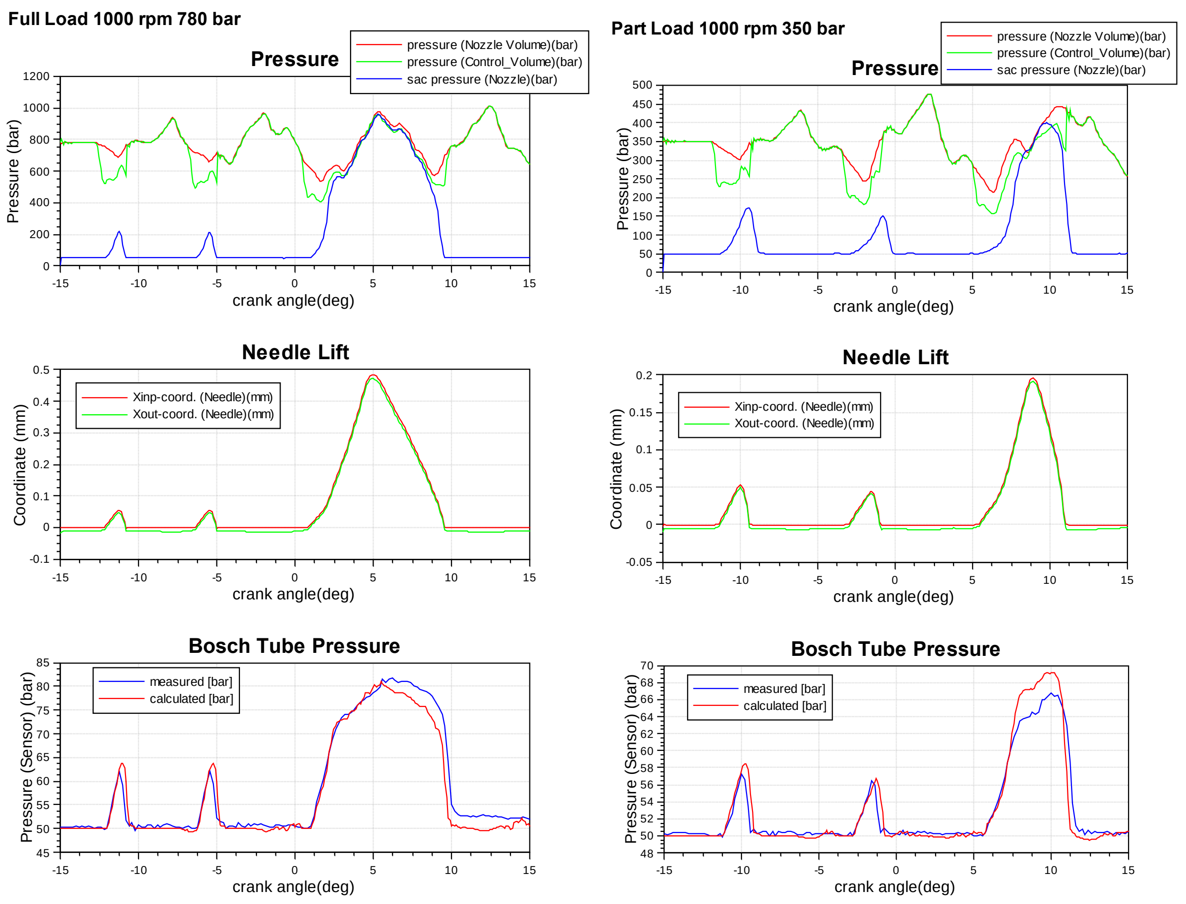


Figure 5-57: Pressures in Injector, Bosch Tube and Needle Lift for two Load Cases

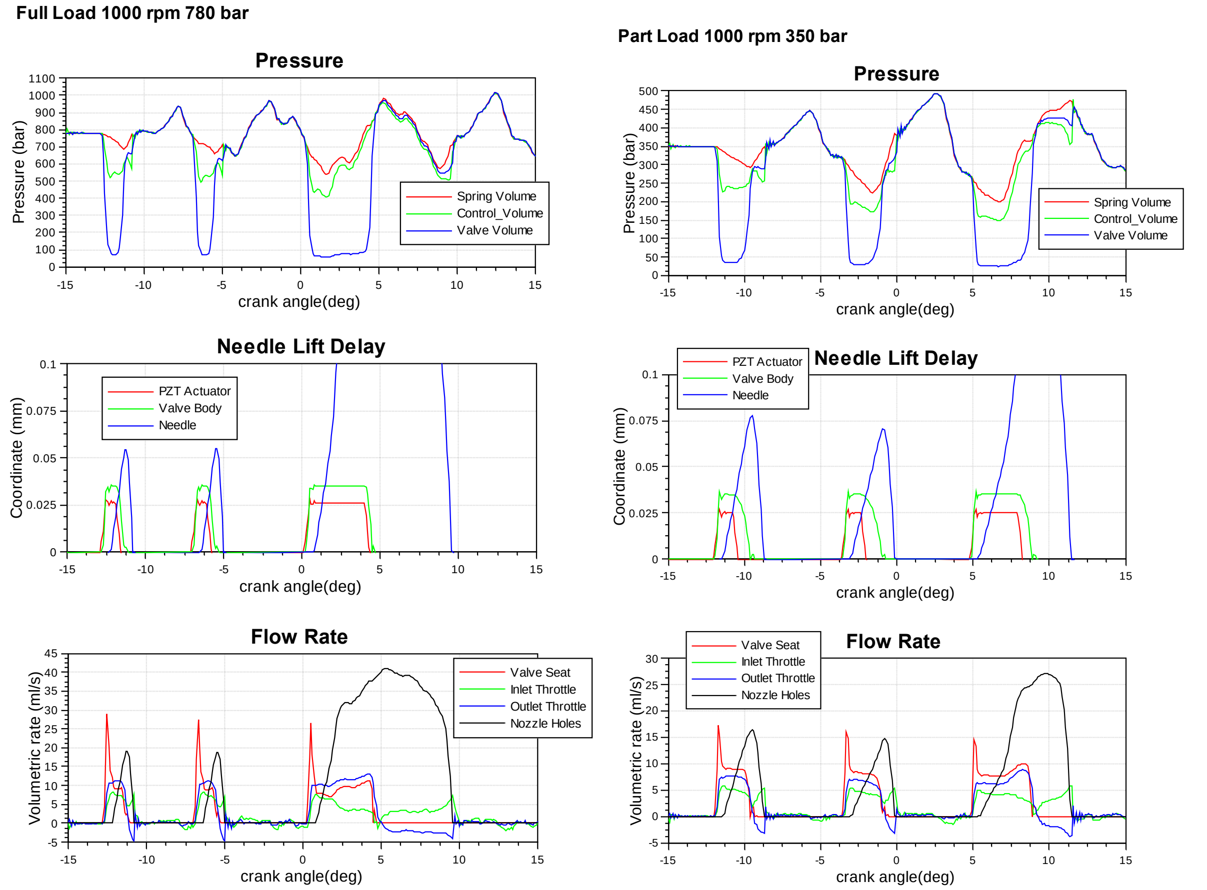


Figure 5-58: Motion of Piezo Actuator, Valve, Needle Lift Delay and Diverse Flow Rates for two Load Cases

The upper graphs in Figure 5-58 show the pressure curves in the spring, control and control valve volumes. Middle graphs represent the lift of the piezo actuator and control valve body with respect to the needle opening/closing ramp. Needle opening delay with respect to the stack actuation is approximately 0.7 deg CA per 1000 rpm or 110 μs. Depending on the rail pressure and actuation voltage, the piezo actuator lift varies from 25 to 32 μm. The control valve lift always stays at 35 μm during the injection. This means that the valve body fully closes the bypass throttle and thus ensures the normal CR-function.

5.5. Further Analysis

A detailed analysis of the calculation results is not carried out in the above examples. The extent of detail in the analysis is determined by the user. For example, it may be interesting to view the variation of the cross-sectional area of the solenoid valves and nozzle orifices, cumulative rate (amount of injected fuel), leakage through piston and needle guide, etc. Moreover, the user is advised to rerun the calculation with different options (e.g. program-calculated nozzle flow area in the common rail example with two-way valve), perform parameter modifications and study their influence on the results. After that the user can invoke the advanced modeling features: optimization (from **Simulation | Search Adjust**) and series calculation with **Case Explorer**.

5.6. GDI System with PID Controller

Common Rail systems are nowadays the most represented fuel injection systems, widely used in diesel engines and gasoline engines. Injection pressure, timing, quantity and rate shape are the decisive factors for proper engine operation. Existing control systems for the rail pressure regulation are mainly based on PID controllers.

This section describes how to use the PID controller element for common rail pressure regulation. This will be demonstrated in a simplified model of a Gasoline Direct Injection (GDI) system shown in Figure 5-59.

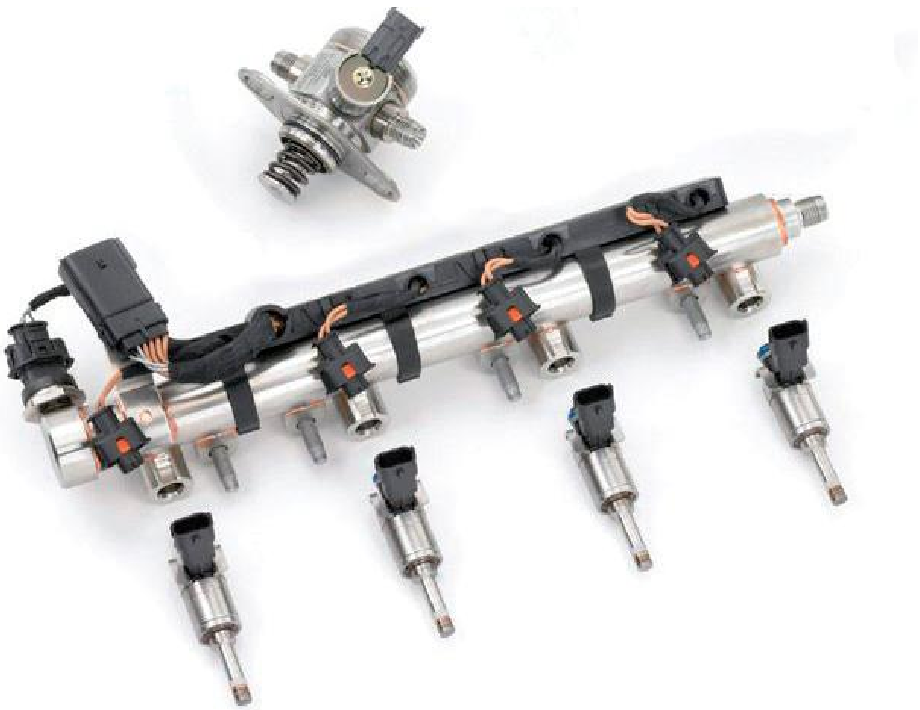


Figure 5-59: GDI Injection System with Common Rail and Four Injectors

5.6.1. Model Description

The simplified model of a Gasoline Direct Injection system is shown in Figure 5-60. It consists of a high pressure fuel pump, common rail, four injectors and PID Controller (electromagnetic pulsating valve). This controller serves for the regulation of the rail pressure at different operating conditions. Within this example, the so-called suction-throttle control of the pump is used, i.e. the pump sucks from the low pressure circuit only the amount of fuel necessary for keeping the rail pressure within the specified range. For this purpose, the PID Controller is connected (by wire connection) to the Suction Throttle of the high pressure pump and Regulator Volume on the common rail. Note that in our example Suction Throttle with PID Controller schematically represent a complex electromagnetic device (fuel metering valve). The rail (Regulator Volume) contains a pressure sensor which sends the actual pressure via the Engine Control Unit (ECU) to PID Controller.

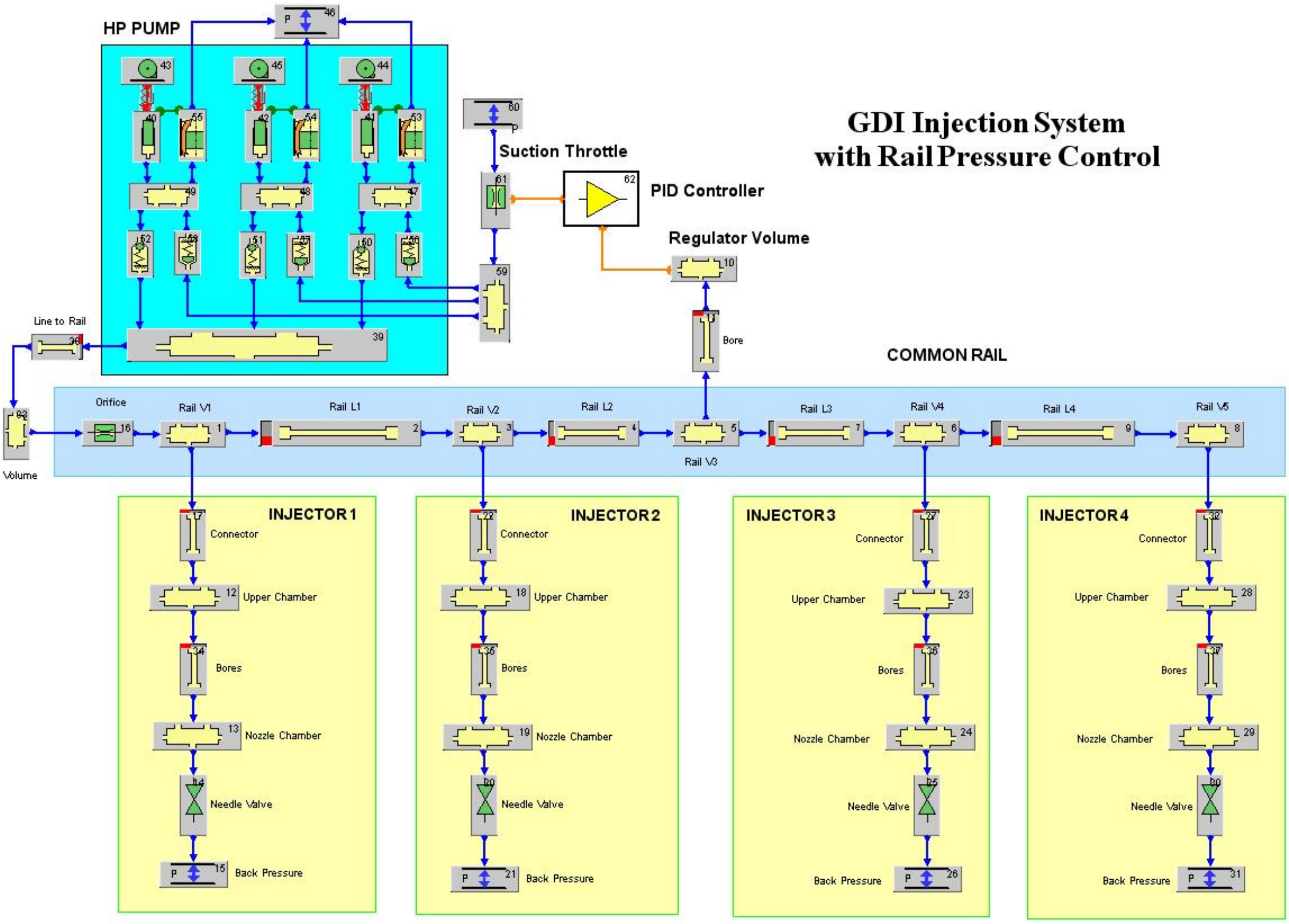


Figure 5-60: BOOST Hydsim Model of GDI Injection System with PID Controller

The fuel metering of the high pressure pump controls the fuel intake volume to the pump depending on the actual rail pressure value (at sensor position) and its guiding value defined in the ECU map. In case rail pressure tends to get lower than the guiding value, fuel flow into the pump is accelerated by increasing the Suction Throttle diameter. If rail pressure tends to exceed the guiding value, fuel flow into the pump is decreased by reducing the Suction Throttle diameter. Note that in reality not the Suction Throttle diameter but the pulse width of the electromagnetic valve is varied. In such a way only the required volume of fuel is supplied to the rail via high pressure pump. The fuel metering valve (Suction Throttle in our case) is actuated by PID Controller.

PID Controller icon is placed between the Regulator Volume 10 (sensor element) and Suction Throttle 61 (actuator element). Suction Throttle area is the controlled variable adjusted by the controller. Regulator Volume contains the pressure sensor.

PID Controller is connected to Suction Throttle and Regulator Volume elements by the wire connections. Wire connection is necessary to enable data exchange between these elements. Connection to Regulator Volume represents sensor channel while connection to Suction Throttle represents actuator channel.

5.6.2. Input Data

In this section only the input data of PID Controller will be discussed.

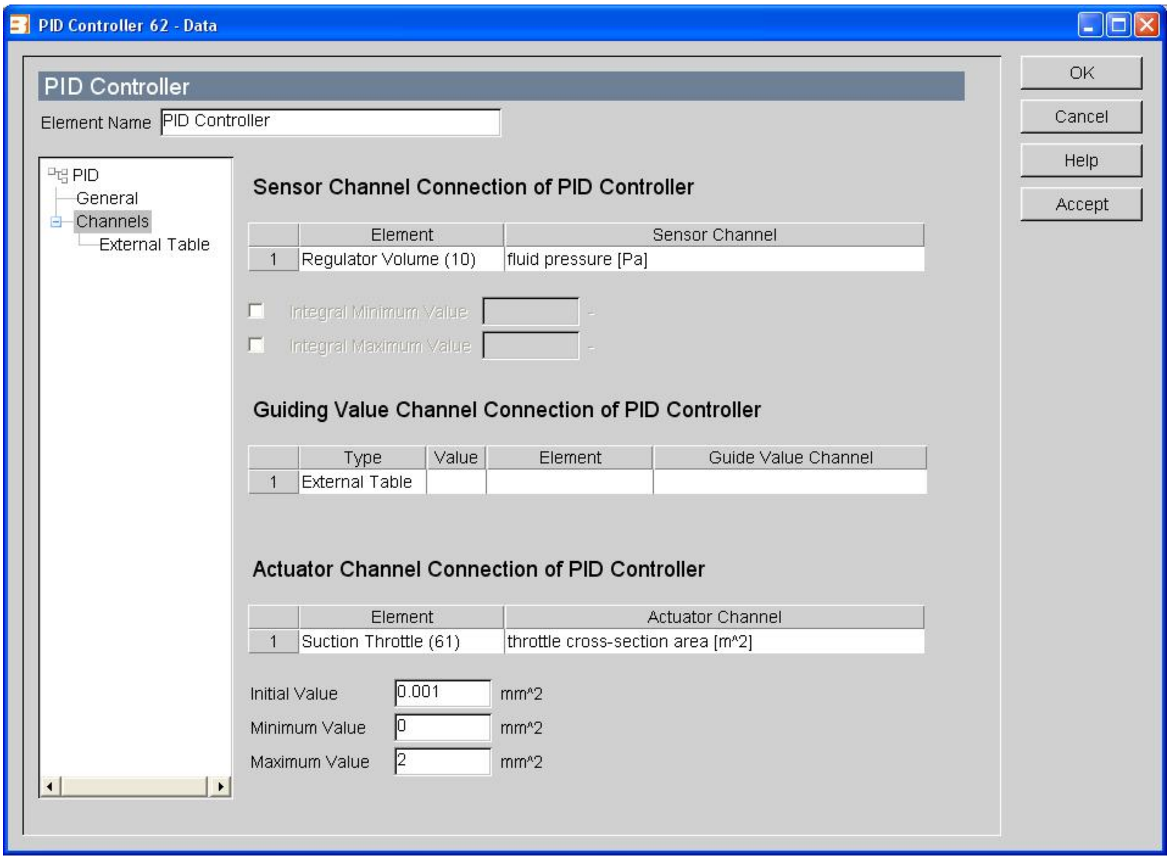


Figure 5-61: PID Controller Dialog (Channels sheet)

PID Controller input dialog consists of two sheets: General and Channels. First we discuss the Channels sheet depicted in Figure 5-61. Basically, this sheet contains the specification of Sensor, Guiding Value and Actuator Channel connections.

Sensor channel in our case is the pressure in the Regulator Volume. Guiding value is the target for the sensor channel value. It can take into account any offset defined in General sheet of properties dialog. Guiding value can be defined as an external (single) value or external table. In our example it is specified in tabular form as shown in Figure 5-62.

Target pressure at the beginning of simulation is set to 6e6 Pa (6 MPa). Table values have to be defined in SI units. At time 0.12 s the guiding value is gradually increased to 7e6 Pa (7 MPa) and at 0.25 s decreased to 5e6 Pa (5 MPa). Note that this is an example definition of the guiding value. It is aimed just to show the capability of PID Controller to properly regulate the pressure in the common rail according to the user-defined function.

Actuator channel (control variable) is the-cross-sectional area of Suction Throttle. Its initial value and minimum and maximum bounds have to be specified. Set initial value to e.g. 0.001 mm^2, minimum value to 0 (throttle closed) and maximum value to 2 mm^2.

In our example interaction between PID Controller and Sensor/Actuator elements occurs not at each time step but at user-specified time intervals. Thus sensor sampling time is set to 1e-6 s while control system response time is set to 5e-7 s (refer to Figure 5-63).

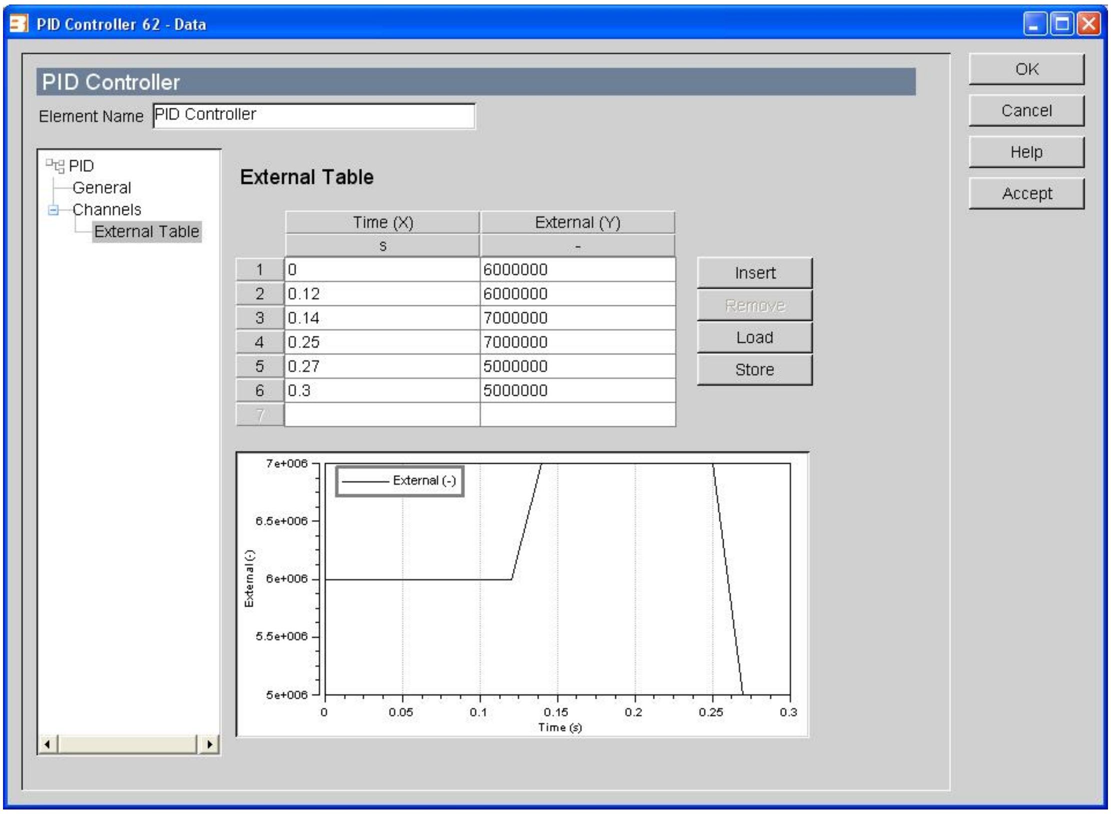


Figure 5-62: Guiding Value of PID Controller (External table)

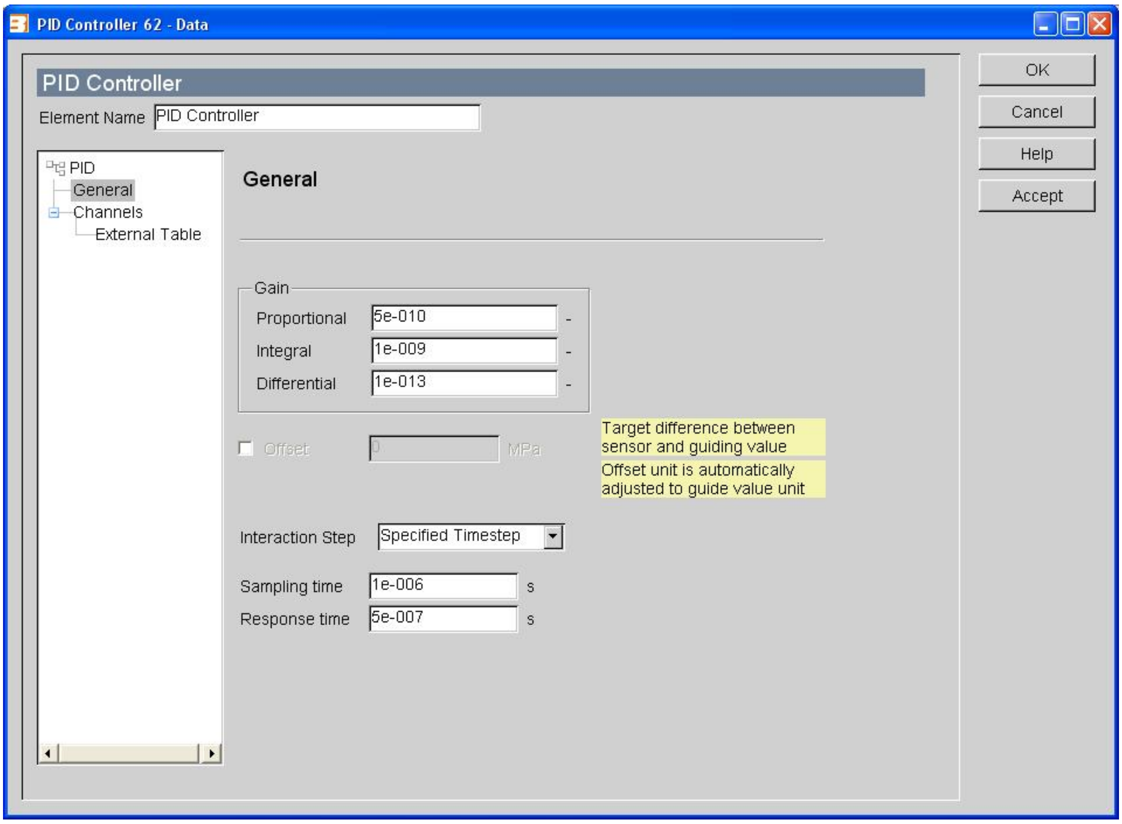


Figure 5-63: PID Controller Dialog (General sheet)

The next, very important step is the tuning of the PID Controller. This, often tedious procedure involves the adjustment of proportional, integral and derivative gains (constants), so that the desired "optimal" system response is attained (in our case target pressure in Regulator Volume). Generally PID controller tuning is a difficult problem, even though there are only three parameters. There exist various simple and more sophisticated tuning methods, such as manual tuning, Ziegler-Nichols method, Cohen-Coon method etc. Their description is beyond of the scope of the primer.

Applying manual tuning, the following "optimal" values were obtained: 5e-10 for proportional gain, 1e-9 for integral gain and 1e-13 for derivative gain.

5.6.3. Calculation Results

Selected calculation results for PID controller are shown in Figure 5-64 and Figure 5-65.

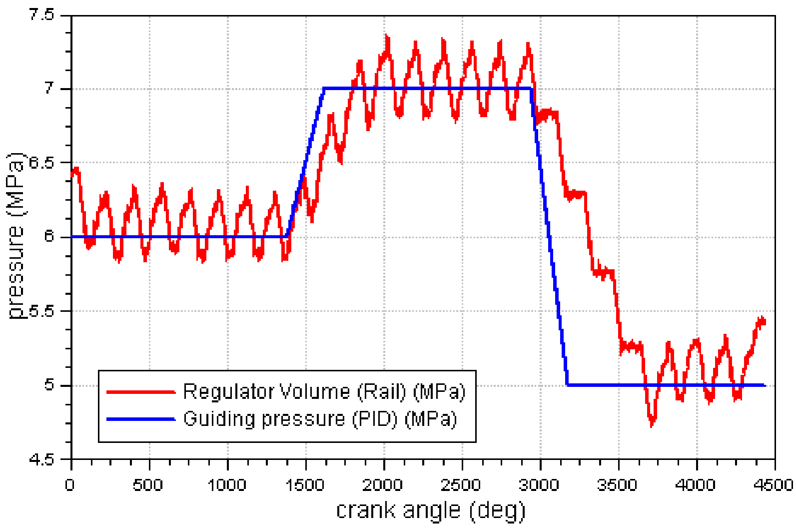


Figure 5-64: Actual and Target Pressure in Common Rail

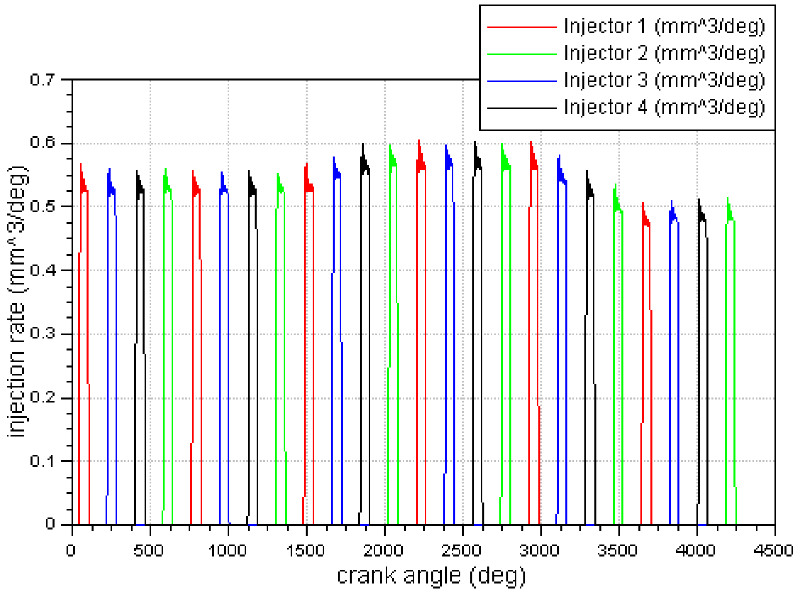


Figure 5-65: Injection Rate through Injectors (firing order 1-3-2-4)

Figure 5-64 shows that the mean pressure in Regulator Volume (common rail) follows the target pressure curve. Of course, the changes of the target pressure at 1440 deg (0.12 s) and 3000 deg (0.25 s) are captured not immediately but with a certain delay because PID controller needs some time to react to the pressure variation and adjust the control variable (maintaining system stability). The injection rates through all four injectors are plotted in Figure 5-65. Obviously at higher rail pressure (7 MPa) between 1500 and 3000 deg the injection rate increases and after that (at lower pressure of 5 MPa) decreases.

Let us now take a closer look at the crank angle interval between 1400 and 1500 deg. Here (at 1440 deg) the target pressure starts rising from 6 to 7 MPa. PID Controller reacts to this at 1450 deg by increasing the throttle flow area to maximum value of 2 mm^2 and keeping it till 1464 degrees (refer to Figure 5-66). The flow rate through Suction Throttle in this range follows the flow area pattern as shown in Figure 5-67.

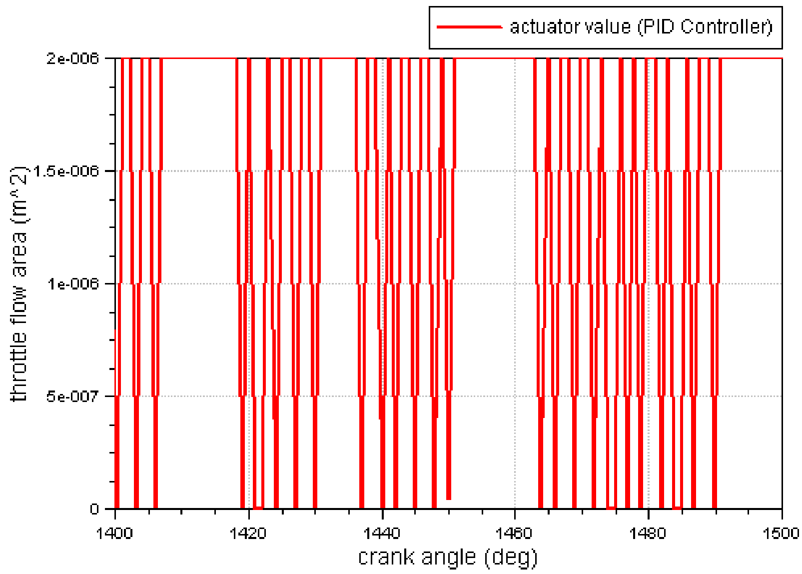


Figure 5-66: Flow Area of Suction Throttle (actuated value)

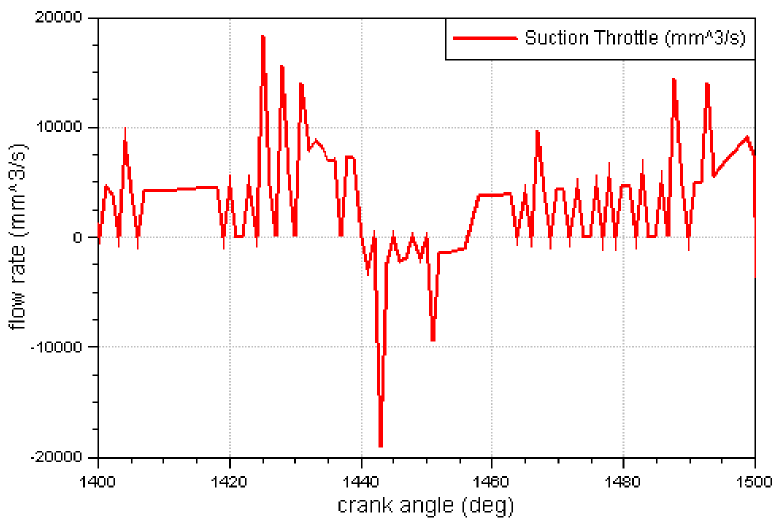


Figure 5-67: Flow Rate through Suction Throttle

5.7. Fuel Injection System for NVH Analysis

This section deals with the fuel injection system model for the NVH (noise and vibration harshness) analysis on cylinder head and engine block. The main target here is the determination of sources for noise and vibration excitation of engines parts. The model chosen for investigation is a common rail system of automotive 4-cylinder inline diesel engine shown in Figure 5-68. The injection system model is divided into three main subsystems: injector, fuel pump and common rail. All three components are modeled as subsystems and finally combined into the complete model of the fuel injection system. Resulting transient loads on the engine parts are injector forces, pressure pulsation inside fuel pump, high pressure pipes and common rail.

Two basic excitation sources associated with the injection process can be defined as:

- impact at contact between mechanical elements inside injector (e.g. injector needle hitting the seat) and eventually fuel pump (mechanical excitaton)
- pressure oscillations in high-pressure components (pipes, volumes etc.) along the fuel injection system (hydraulic excitation).

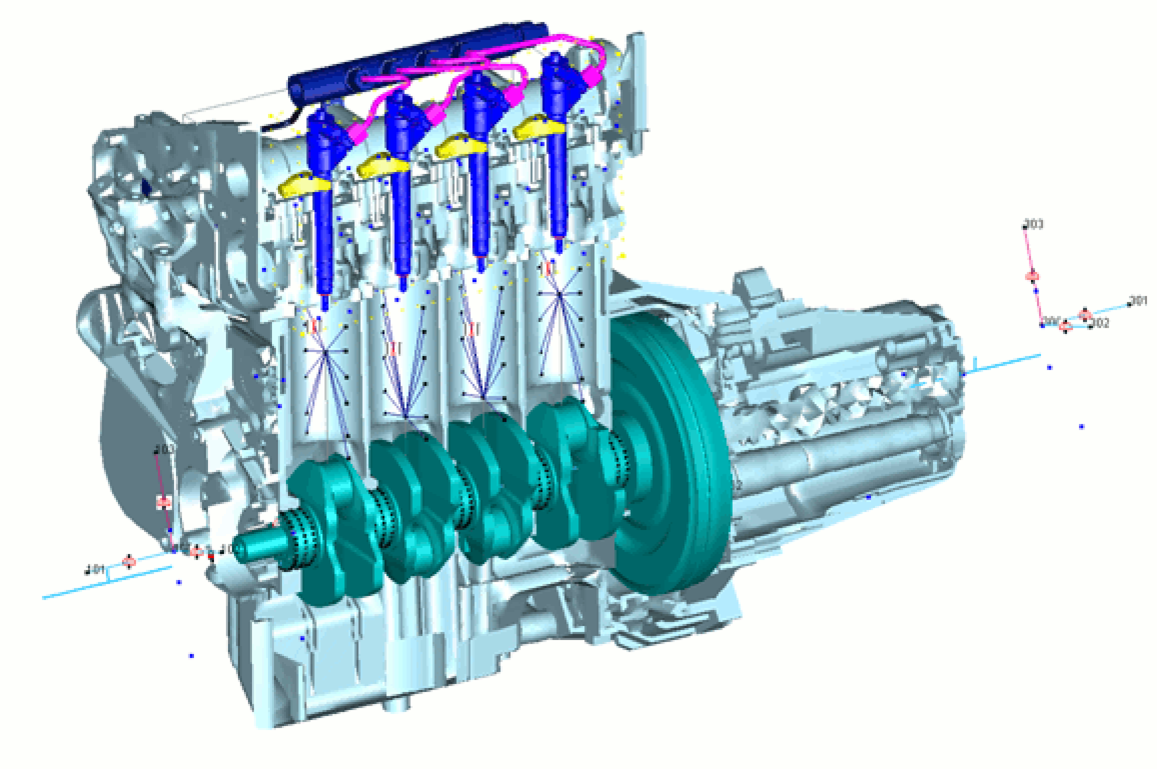


Figure 5-68: Common Rail and four Injectors mounted on Cylinder Head

5.7.1. Model Description

HYDSIM model of the complete fuel injection system with hydraulic, mechanical and control elements is simulated. Fior modeling, the system is split into individual groups: high-pressure accumulator (common rail), four independent injectors (refer to Figure 5-69) and high-pressure pump with PI controller (refer to Figure 5-70). The PI controller element simulates the so-called inlet-suction (metering) control of the fuel pump. Based on it, the pump sucks on inlet the amount of fuel just necessary to maintain specific pressure level in the common rail (depending on load, speed etc.).

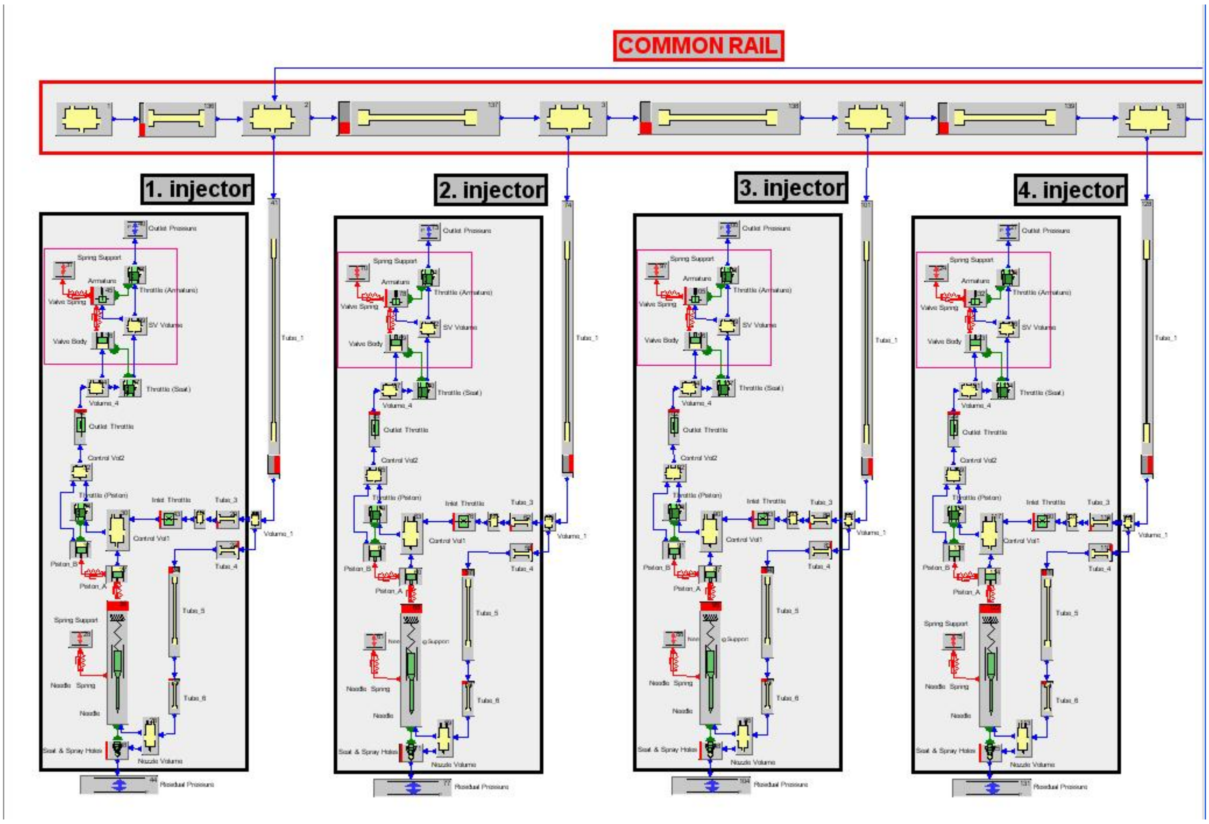


Figure 5-69: HYDSIM Model (part 1): Common Rail with Four Injectors

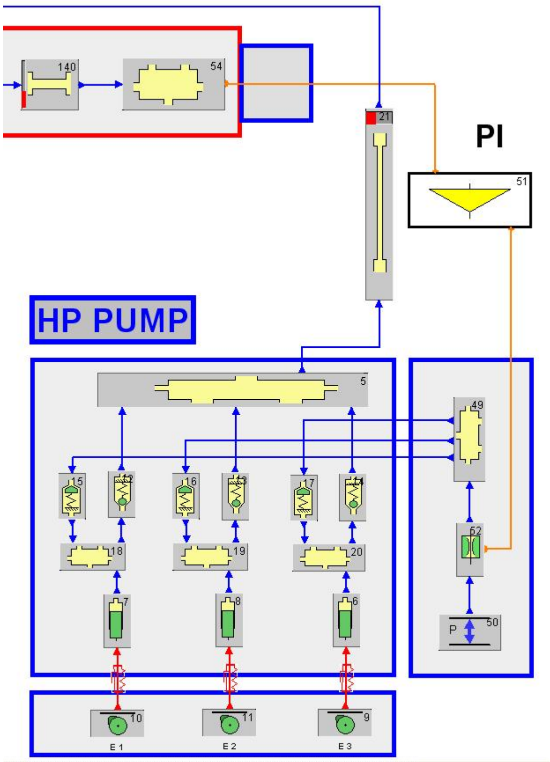


Figure 5-70: HYDSIM Model (part 2): Fuel Pump with PI Controller

5.7.2. Excitation Forces from Injection System

The goal is to predict and understand the impact of fluid flow excitations from the FIS on the noise and vibration in the engine under different load conditions. The noise from a diesel engine is composed of many components emitted from different sources.

The common rail injector consists of multiple hydraulic and mechanical components. Volume as hydraulic element is filled with compressible fluid subjected to high-pressure oscillations. If a piston is moving in a volume, fluid inside volume exhibits highly dynamic force on the pressurized area of the piston. This force will be transmitted to connected mechanical elements and eventually to the injector housing as soon as the contact between piston and housing occurs. On the other side, unbalanced reactions force will act on the volume walls. For volumes that do not contain piston-type elements, the pressure is assumed to be equally distributed inside the volume. Thus the resultant integral force on the volume walls is zero.

Mechanical components generate excitation forces at their impact with the injector housing. Typical example is needle element regularly (at each injection event) hitting the housing at seat and stop positions. During impact with any of them, the needle generates the force on the housing. Many mechanical elements like needle, control piston solenoid armature are connected with housing via mechanical springs. These springs transmit excitation forces to the housing as well.

At each calcualiton step HYDSIM calculates pressure and flow rate along the hydraulic system and displacements and velocities of all mechanical elements. Based on these state variables, the program calculates excitation forces on the housing shown in Figure 5-71.

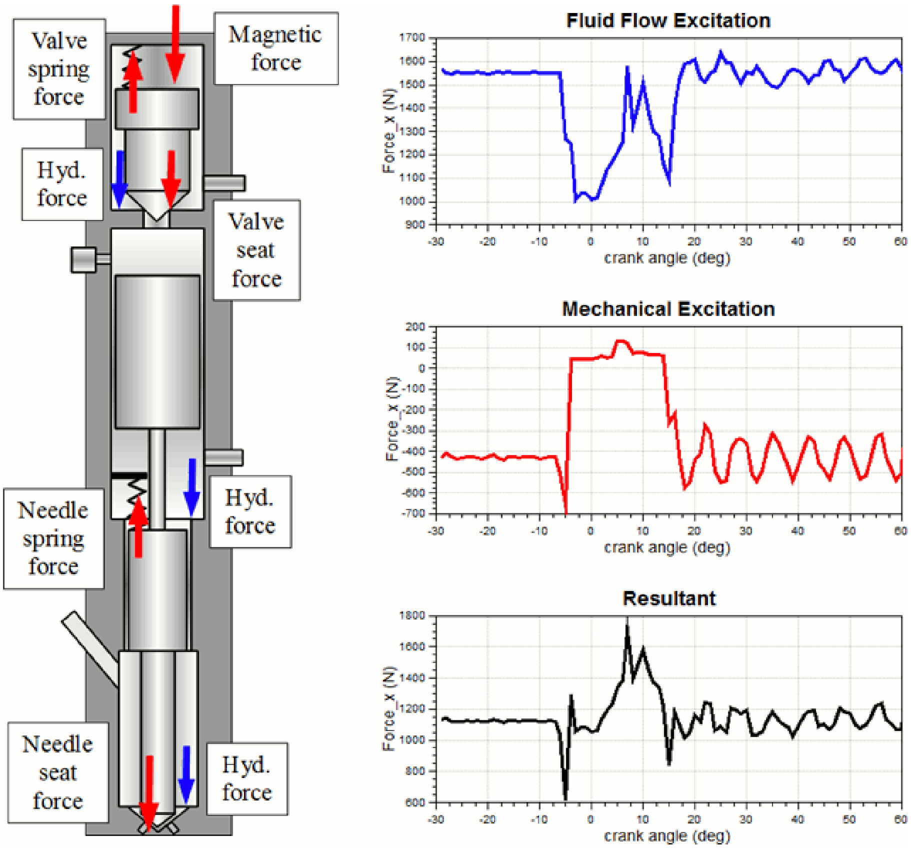


Figure 5-71: Hydraulic, Mechanical and Resultant Force from 1st Injector

The excitation foces calculated by HYDSIM are usually applied on the specific nodes (degrees-of-freddom) of the Finite Element (FE) model of e.g. cylinder head. Often a large number of HYDSIM elements with active force calculation correspond to one single node in the FE model of the cylinder head. For this case a macro-element option is available: the user can selects the desired elements/area in HYDSIM model and define them as a macroelement (clicking right-mouse button). Usually this macroelement refers to one node in the respective FE model. For all elements belonging to the macroelement (e.g. complete injector, refer to Figure 5-72) the excitation forces will be calculated and summed up. The resulting forces will be then stored onto an ASCII file and can be directly applied on the specific node. To calculate and export excitation forces for a specific element, the check box **Link to Excite Power Unit** (**Excite PU** Link) has to be activated in the **Properties** dialog.

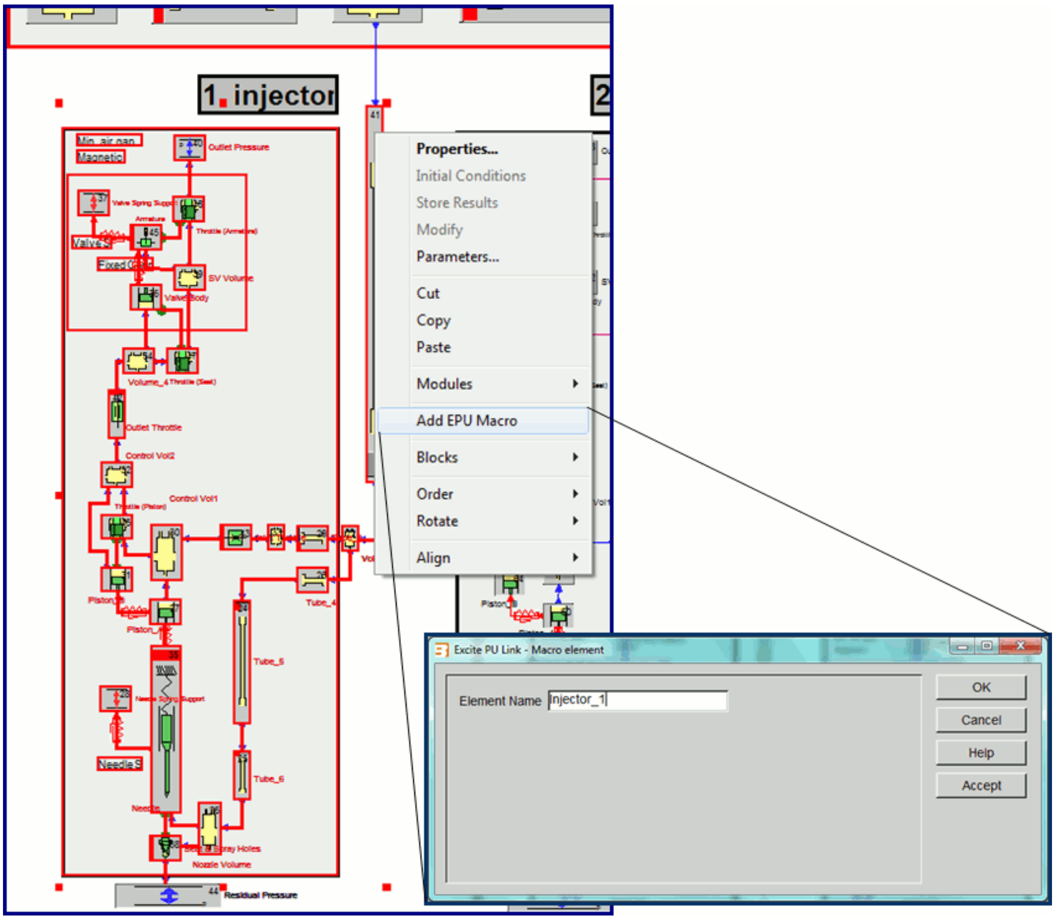


Figure 5-72: Hydsim Macroelement (1st Injector)

Obviously the calculation procedure of the excitation force is different for the specific element. For e.g. Cam Profile element separate forces on the camshaft are calculated in three directions as shown in Figure 5-73. Apart from the injector-generated forces, our simulation model provides excitation forces on the flange and driving shaft of the high-pressure pump in three directions (0, 120 and 240 deg). Force values are stored in the GIDas (ASCII) file format shown in Figure 5-74.

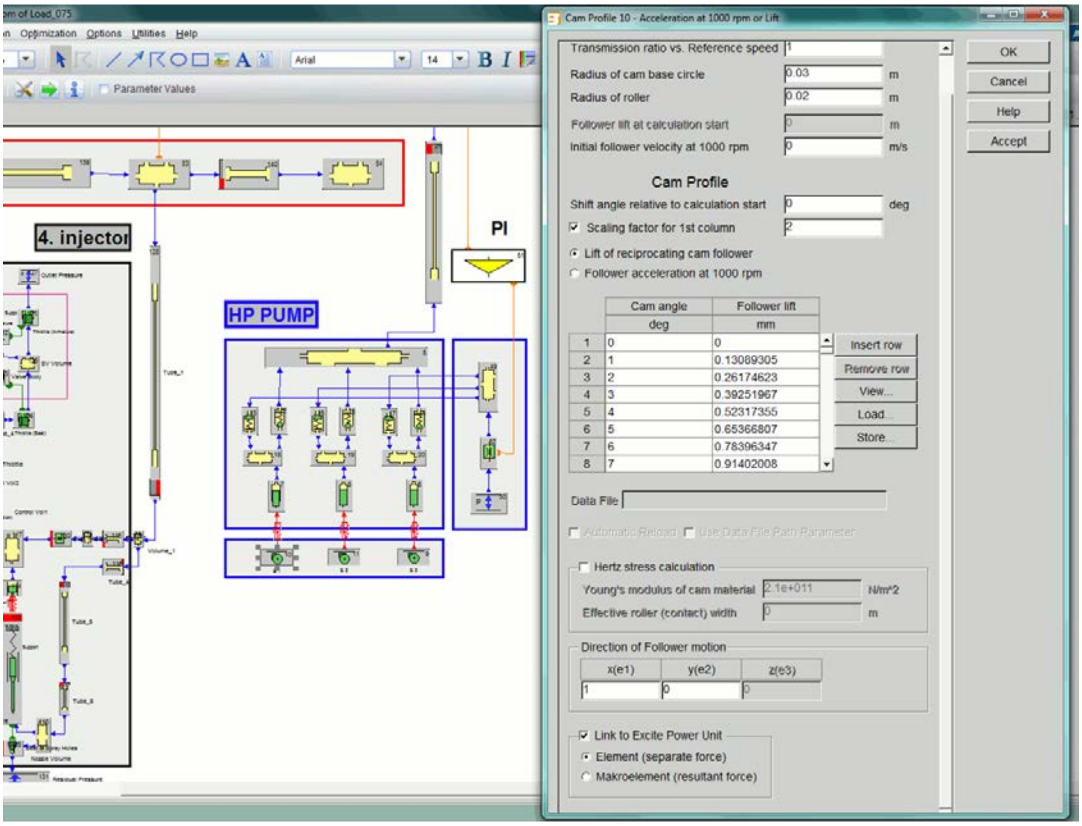


Figure 5-73: CAM element of HP Pump with active Excite-PU Link

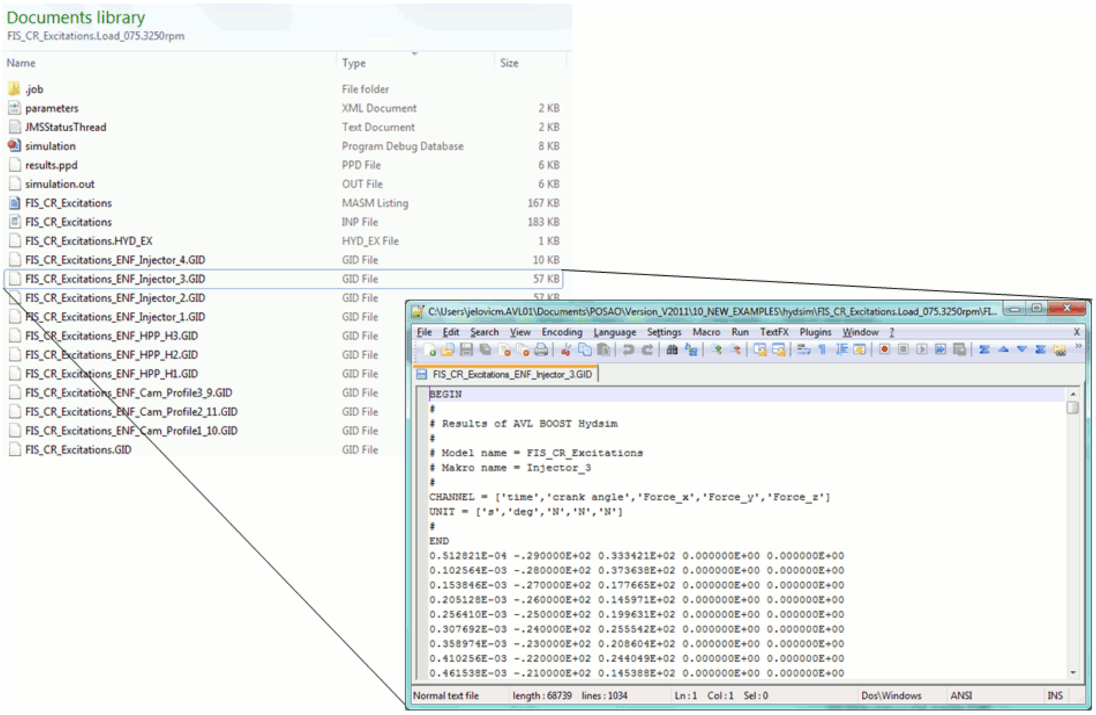


Figure 5-74: Excitation Forces stored on GIDas File in Working Directory

5.7.3. Calculation Results

Typical excitation forces for NVH analysis in our example are injector housing forces and forces on the flange and shaft of the high-presure pump. These forces are plotted in crank angle domain in Figure 5-75. One can observe that the frequency content of the injector forces and pump forces is completely different. Depending on the system properties and operating conditions, it can vary from 300 to 3000 Hz.

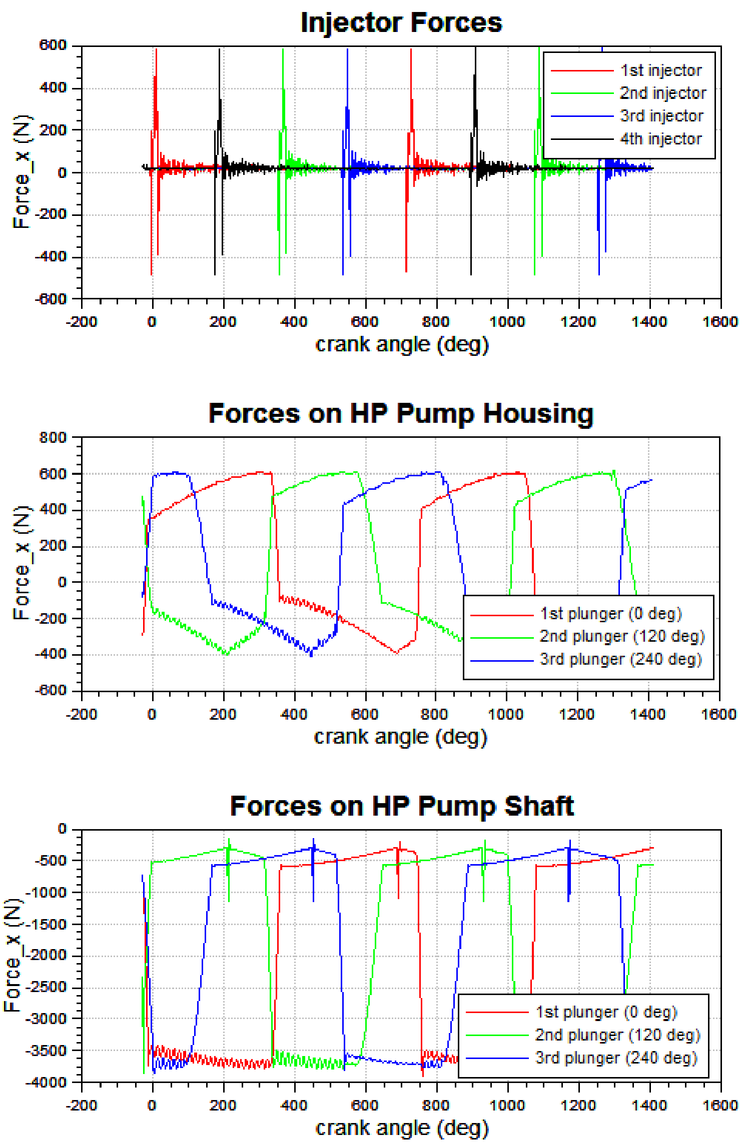


Figure 5-75: Excitation Forces from Fuel Injection System

6. CAM DRIVEN SYSTEMS

This chapter is aimed at introducing the cam-driven injection systems: a typical electronically-controlled unit injector (combined pump-nozzle unit) and a conventional pump-line-nozzle system in both mechanical and electronically-controlled forms. We begin with the model of a typical electronic unit injector with a solenoid control valve.

6.1. Unit Injector

The schematic of the electronic unit injector (pump-nozzle unit) is shown in *Figure 6-1*. It consists of the following main parts: Cam Profile 1, Plunger 2 with a spring, Injector Holder 3, Outlet 4, Nozzle 5, Solenoid Valve 6 and Inlet 7. The injection timing is controlled by the solenoid switching, which is independent of the crankshaft rotation angle. As long as Solenoid Valve 6 stays open, Plunger 2 pumps the fuel into the outlet. When Solenoid 6 switches off (closes Inlet Line 7), pressure in the nozzle volume (under the needle guide) rapidly increases due to Plunger 2 motion. As soon as the nozzle cracking pressure is reached, the needle opens and the fuel is injected into the cylinder. The duration of the closed phase of the Solenoid 6 determines the injection quantity (cumulative rate).

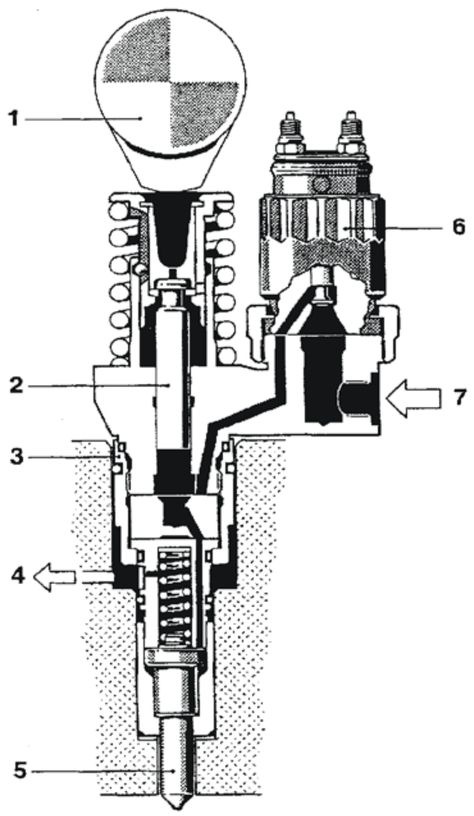


Figure 6-1: Schematic of an Electronically-controlled Unit Injector

6.1.1. Creating the Model

The BOOST Hydsim model of the unit injector is shown in *Figure 6-2*. It consists of the following elements:

- Cam Profile 1[6] **(CAM/Cam Profile)**
- Plunger 2 **[PUMP/Plunger]**
- Pump Chamber 3 **(VOLUME/Standard)**
- Solenoid Bore 4 **(LINE/Laplace)**
- Solenoid Volume 5 **(VOLUME/Standard)**
- Solenoid Valve 6 **(THROTTLE/Time-controlled)**
- Nozzle Holder 7 **(BOUNDARY/Mechanical)**
- Nozzle Bore 8 **(LINE/Laplace)**
- Nozzle Chamber 9 **(VOLUME/Standard)**
- Back (Leak) Pressure 10 **(BOUNDARY/Pressure)**
- Inlet Pressure 11 **(BOUNDARY/Pressure)**
- Cylinder Pressure 12 **(BOUNDARY/Pressure)**
- Nozzle Orifice 13 **[NOZZLE/SAC(Basic model)]**
- Needle 14 **[NEEDLE/Standard(Obsolete model)]**

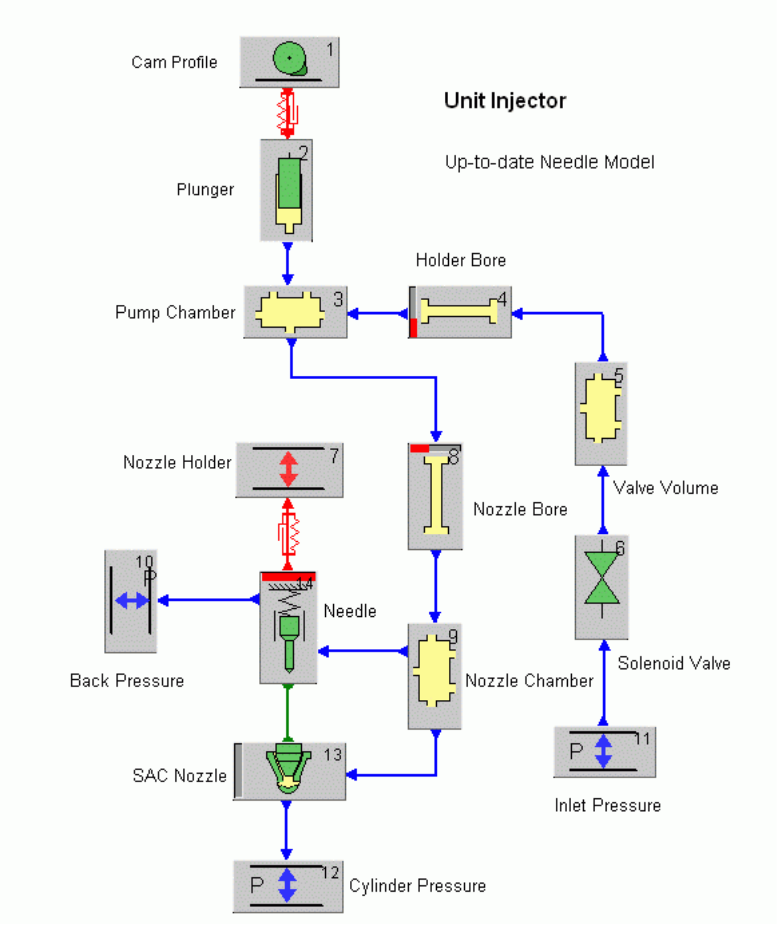


Figure 6-2: BOOST Hydsim Model of a Unit Injector

[6] Numbers given for elements may differ from those in your model. Reference *Figure 6-2* for numbered items.

The system is relatively compact. The user is already familiar with most of the elements from the common rail examples. However, there are two new elements in the system: Cam Profile and Plunger. Plunger geometry is very simple in this particular case (without a metering ramp or control helix) and its dynamic model is identical to that of a piston.

Previously, the procedure for creating a BOOST Hydsim model was described: placing the model icons and connecting them with appropriate connections. This chapter concentrates only on the new features in the system.

6.1.1.1. Cam Profile

The very first element of the model is Cam Profile. Here it plays the role of a mechanical boundary condition (external excitation). Cam Profile must not necessarily be the boundary element of the system: it can be further connected to the SOLID/Shaft element, which is aimed to model the part of the driving shaft (camshaft). In this case, however, the modeling becomes more tedious and therefore will not be discussed at this stage. The input dialog of the Cam Profile element is as follows:

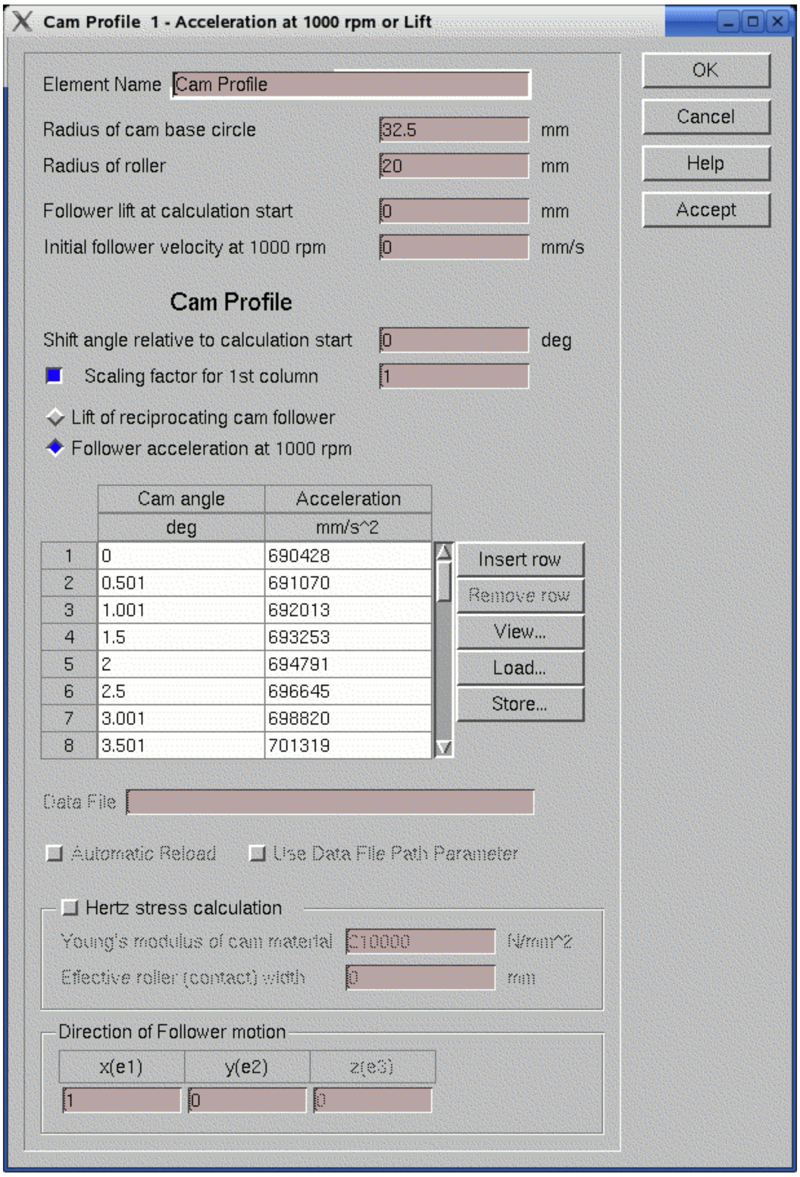


Figure 6-3: Cam Profile Input Dialog

The user has to provide the table of the acceleration values of the reciprocating cam follower at 1000 rpm (as a function of cam angle). Alternatively, the lift of the cam profile contact point with the follower in x direction can be specified. Follower lift as input data must be used with care. If the lift data is not precise enough, the numerical derivatives (velocity and acceleration) may contain coarse errors and the calculation results will be incorrect. The follower velocity and acceleration curves need to be checked carefully in each calculation.

For a constant camshaft speed ω, the time acceleration (second time derivative) of the follower is related to the angular acceleration (second kinematic derivative) by the formula

$$\ddot{y} = y''\omega^2, \tag{6.1.2}$$

where $\ddot{y} = \dfrac{d^2 y}{dt^2}$, $y'' = \dfrac{d^2 y}{d\varphi^2}$ and $\omega = \dfrac{\varphi}{t} = const.$

In the input dialog, the user has also to specify the element name (default name is Cam Profile), radius of cam base circle and of the roller. In this example (Cam Profile — boundary element), these radii are not necessary for the dynamic calculation as the follower acceleration is taken directly from the table (for a roller follower the acceleration of the roller center is implied). If the cam profile has no input connection to a shaft, radii of cam base circle and the roller are required only for the calculation of pressure angle, normal force and camshaft torque, otherwise they can be omitted. Furthermore, the user has to provide initial conditions (lift and velocity) of the follower motion at calculation start (if these are not zero). For a lift table, only initial velocity is required. Additionally, a shift angle relative to calculation start can be defined. Positive shift implies that from the calculation start up to the shaft rotation angle less than specified Shift angle, the follower (roller tappet) rests on the cam base circle. Hence, if positive Shift angle is defined, initial conditions of the follower must be zero.

For all Cam Profile elements, the user also has the option to calculate the Hertz contact stress between fuel cam and roller follower. To do this, activate **Hertz stress calculation** and input the required Young's modulus for the cam material (default value 210000 N/mm^2 for normal steels). Input also the **effective roller width** (deduct any roller edge radii or undercuts from the total width).

6.1.1.2. Plunger

The input dialog of the Plunger element is shown as follows:

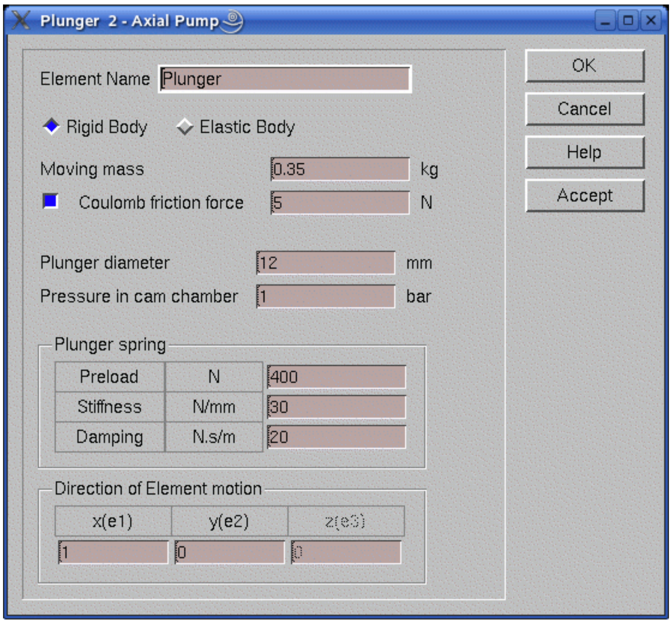


Figure 6-4: Plunger Input Dialog

Here, specify the moving mass (plunger mass + roller tappet mass + 33% of plunger spring mass), plunger diameter, Coulomb friction force (if any), pressure in the cam chamber and plunger spring parameters (pre-load, stiffness and damping).

The mechanical connection (red line) between Cam Profile 1 and Plunger 2 represents the linearized stiffness and damping of e.g. the cam drive and not the plunger spring data. The plunger spring data is specified in *Figure 6-4*.

6.1.2. Calculation Control

Simulation | Control

Before starting the calculation, the user has to supply all necessary input data, define fluid properties, initial conditions and desired output and calculation control data. All input data of this example is not provided here. It can be found in the example file `../BOOST Hydsim/v2011/example/unit_injector.hyd` supplied with the software, so the user can easily load it and copy all necessary data.

☞ **Note:** For cam-driven systems containing Cam Profile or Cam Plate element, **Reference Angle** must be chosen as **Simulation Domain** in the **Calculation Control** dialog. In this case the **Reference speed** has to be specified, which is the cam rotation speed (refer to Figure 6-5).

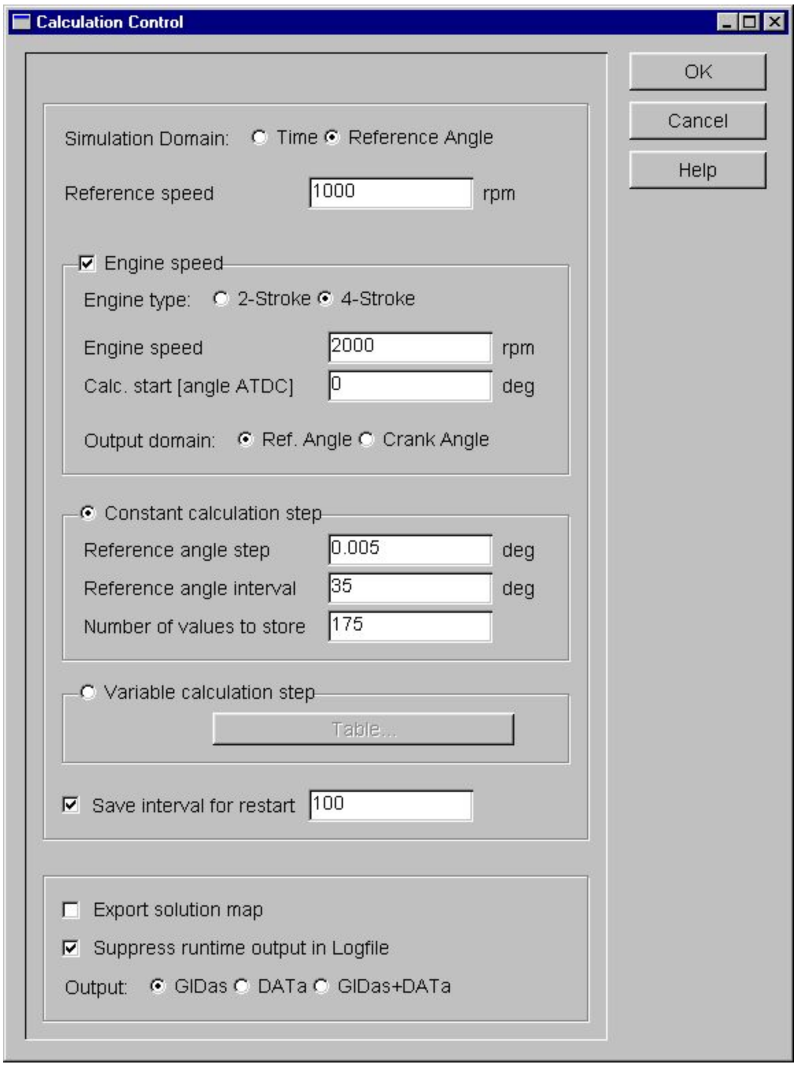


Figure 6-5: Calculation Control Dialog

As stated earlier, for the Reference angle domain the specification of the reference speed is mandatory, otherwise an error will be issued at calculation start. Reference speed here implies the running speed of the camshaft (1000 rpm in our case).

The **Reference angle step** is a fixed step for a constant **Reference speed**, but has to be adjusted if the reference speed is varied during modeling. Internally BOOST Hydsim converts the reference angle step to the time step which is important for integration. **Time step** must be defined within certain limits. If the step is too large, the calculation may be unstable, if too short it may cause excessive calculation time.

☞ **Note:** There is no universal rule for definition of the time step. It depends on the specific system. However, for the high pressure dynamic systems typical time step range is: $\mathbf{10^{-8}\ s < \Delta t < 10^{-6}\ s}$. Values outside this range should not be used in standard applications. Especially the larger value, below 10^{-6} seconds, can be critical and cause numerical instability.

Next, the **Reference angle interval** must be specified. This is the period over which the calculation is to be performed. Clearly it has to be large enough to cover the rise in cam profile (to maximum lift) and the switching times (closure phase) of the solenoid valve. For each reference speed value, the switching time instant and duration of the solenoid has to be adjusted appropriately, to suit the required injection timing (start of injection) and injection quantity (duration of injection).

Number of values to store is the total number of calculated points in the required calculation duration. A minimum of five points per cam degree is a normal minimum. Ten points per degree is better but the larger the number, the longer the calculation time.

> ☞ **Note:** In this example, first we perform a standard calculation till 35 deg cam angle (refer to **Reference angle interval** in **Calculation Control**). After that we will perform the second part of calculation from 35 to 60 deg using **Restart** option (refer to *Section 6.1.5*). For this, it is necessary to switch on **Save interval for restart** button and input desired interval**.** This interval must be smaller than/equal to **Number of values to store.** At each **Save interval for restart** input**,** BOOST Hydsim will generate a special output file <model_name>.STA file for **Restart** calculation (refer to *Section 6.1.5*).

The initial conditions (**Element | Initial conditions)** should be specified in such a way that the system stays in equilibrium at the starting point (zero angle). For example, if the inlet pressure is set to 6 bar, then the initial pressure in Pump Chamber, Solenoid Volume and Nozzle Chamber has also to be 6 bar and the initial cam lift has to be zero. Of course, the user is free to specify other initial conditions if these are relevant for a specific application.

6.1.3. Running the Calculations

To perform the calculation please refer to *Section 4.4*.

After loading a `results.ppd` file in the IMPRESS Chart window, the user may observe that the results can be plotted in two domains: Time and Reference Angle. This is always the case if the shaft angle domain is chosen for calculation because the integration is performed in time domain anyway. It is often convenient to use the cam rotation angle as a reference domain.

Typical calculation results for a unit injector would be pressures in volumes (Pump Chamber 3, Solenoid Volume 5 and Nozzle Volume 10) as well as lift of Needle 14 and injection rate through Nozzle Orifice 13. As maximum needle lift is 0.3 mm, the sac pressure always stays lower than nozzle chamber pressure. These results are shown in *Figure 6-6* and *Figure 6-7*, respectively. Their analysis is not carried out here but is left for the user. The user is free to choose other output data for analysis. Any desired output parameters (if selected in Element/Store Results) can be viewed with the post-processor Impress Chart. Refer to *Section 4.5* for inputting results data in the Impress Chart window.

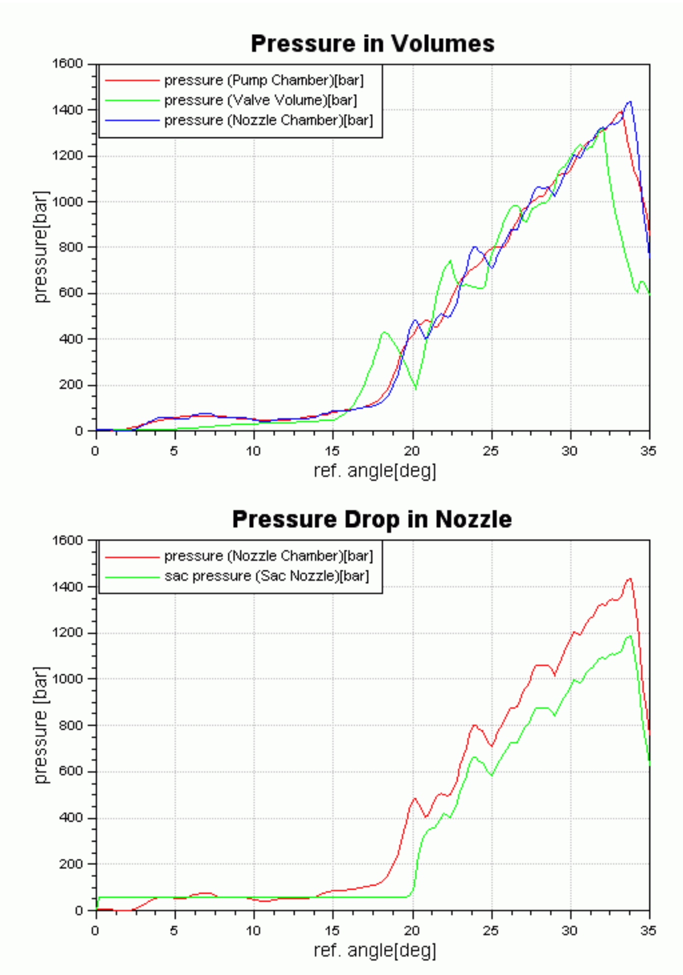


Figure 6-6: Pressure in Volumes and Pressure Drop across Needle Seat

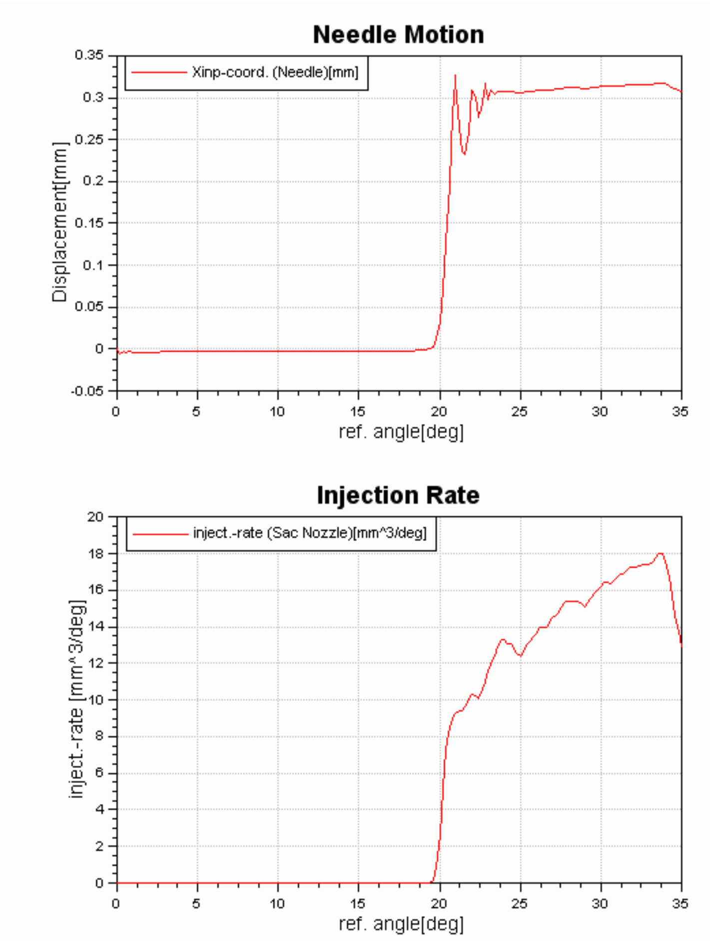


Figure 6-7: Needle Lift and Injection Rate

☞ **Note:** Switch the X-axis between **Time** and **Reference Angle** domain, by highlighting the appropriate output parameter, scroll down the menu until X-axis domain appears (*Figure 6-8*), then choose desired domain.

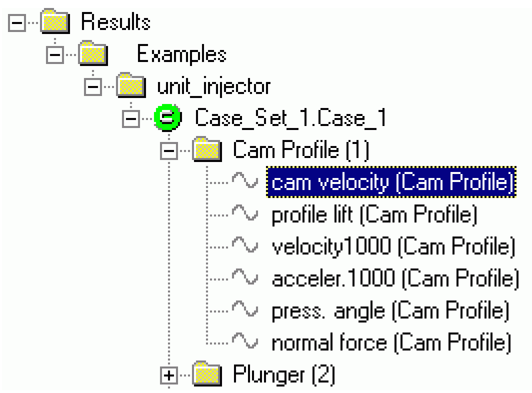


Figure 6-8: X-Domain Menu

☞ **Note:** Add output parameters (from an expanded results tree) to the X-axis domain menu by first highlighting the appropriate parameter and clicking the right mouse button. From the Pop-up menu select **Use as X-Axis** and the selected parameter will be added to the X-axis domain menu list (*Figure 6-9*).

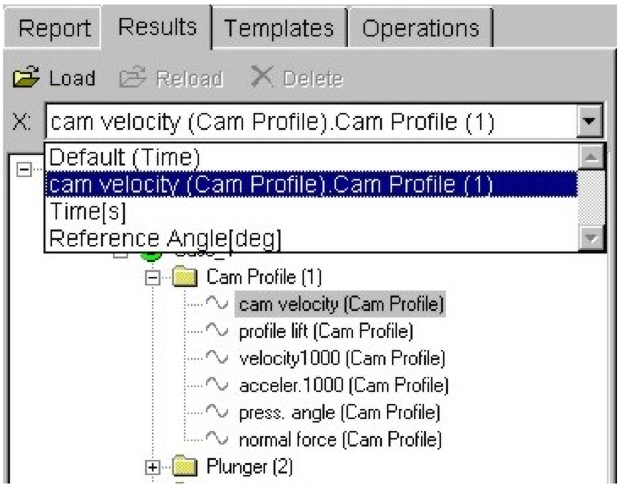


Figure 6-9: X-axis Menu List

6.1.4. Running Calculation with Crankshaft Angle as Output Domain

In the previous example, Reference Angle represents the cam rotation angle from the start of roller follower lift (pumping plunger lift in this case, since there is no rocker drive) - this is for the element Cam Profile 1.

In some cases, in examples like this one where the Reference Angle is the Simulation domain, the user would like to plot results in terms of Crank Angle.

This is feasible through the Calculation Control Dialog. It is necessary to activate the check button in the front of Crank Angle for Output domain and enter Calculation start [the angle ATDC-After Top Dead Center]. In our example, it is set to –40 deg (refer to *Figure 6-10*). Crank Angle will be added to the x-axis domain menu list.

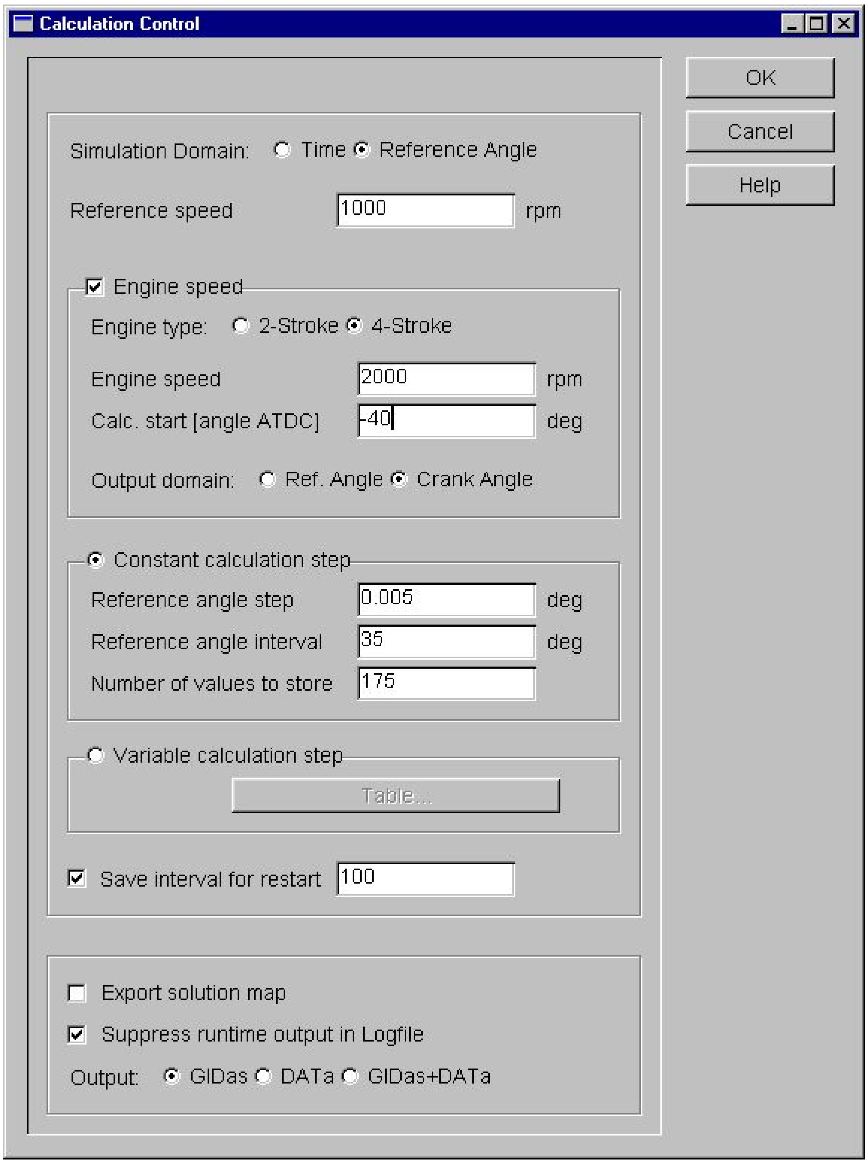


Figure 6-10: Calculation Control Dialog with Crank Angle as Output Domain

Save this change under a different file name, e.g. `unit_injector_ca.hyd`, and restart the calculation. Refer to *Sections 4.4* and *4.5* for running calculation and inputting results data in the Impress Chart window.

Switch x-axis domain by means of x-axis menu list (*Figure 6-9*) and plot graphs.

In *Figure 6-11* two layers are shown with different x-axis domain. The upper diagram has Crank angle as the x-axis domain while the lower one shows the same output parameter but with Time as the x-axis domain.

The user may observe that the starting angle (-40 deg), on the graph with Crank angle as x-domain, is the angle from **Calc. start [angle ADTC]** defined in the Calculation Control dialog. The Crank angle interval is 70 deg, which is twice the Reference (Cam) angle interval of 35 deg.

Output parameters, which are plotted, are chosen the same as in the upper graph in *Figure 6-6*.

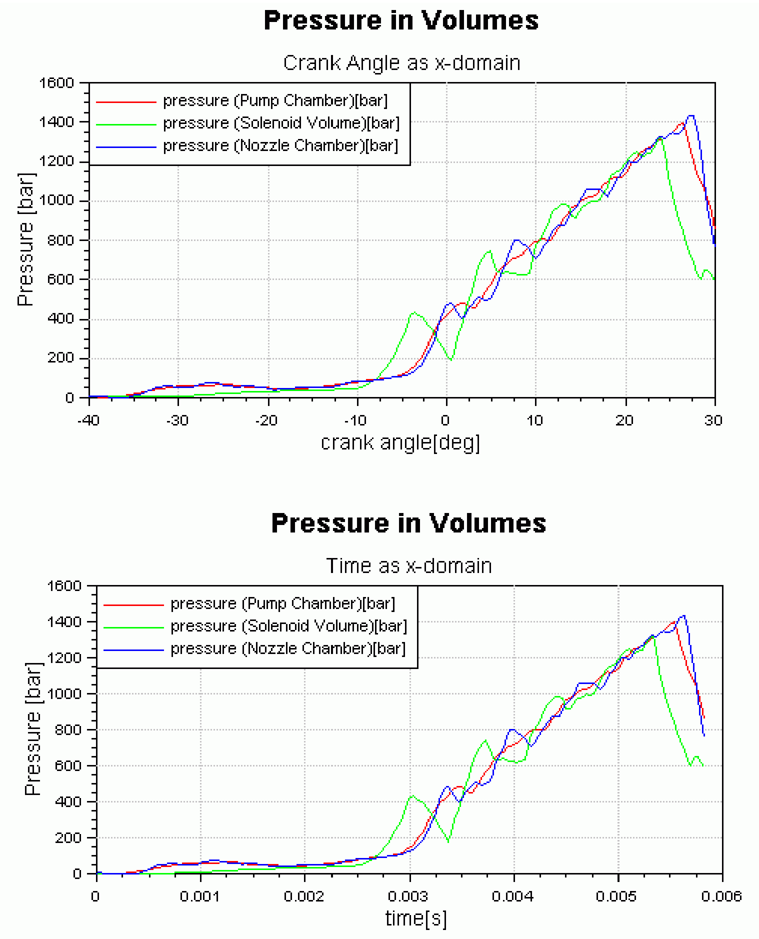


Figure 6-11: Crank Angle as X-axis (upper graph)

6.1.5. Restart Simulation

To perform **Restart** simulation, it is necessary to make changes in the **Calculation Control** dialog. Set **Reference angle interval** to 60 deg, activate Restart Calculation button and deactivate **Save interval for restart**. The Restart option uses the previously started and saved calculation, i.e. the results will be appended to the previous GIDas file.

☞ **Note: Restart** calculation uses model_name.STA file, so it is necessary to perform a standard **Run** beforehand (to create model_name.STA file). For **Restart,** the calculation interval has to be extended (refer to **Reference angle interval** in *Figure 6-5*) and the model name preserved.

6.1.6. Running Restart Simulation

If no errors occur, this window should look as follows by the end of calculation:

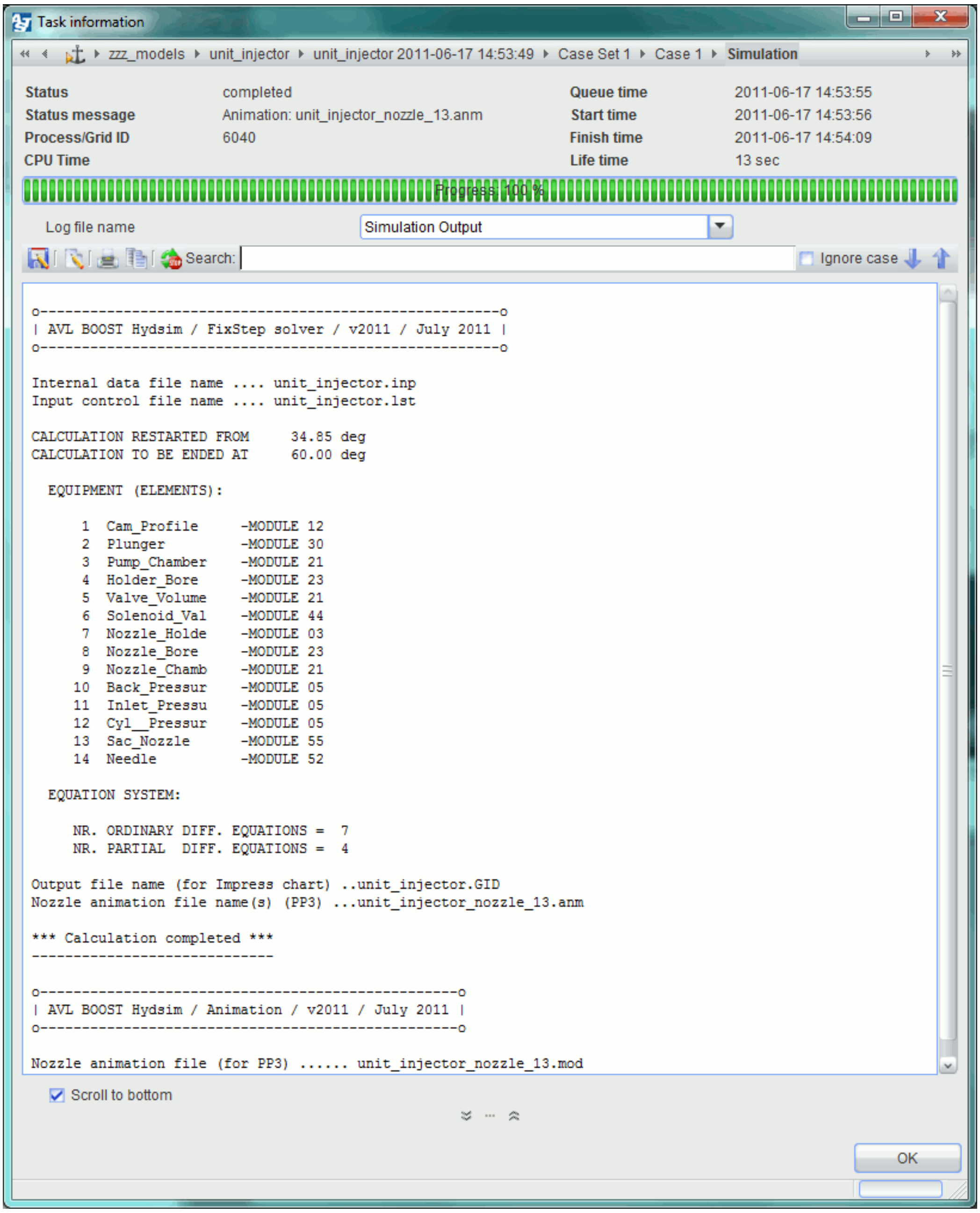


Figure 6-12: Task information of Restart Simulation

In the already created Impress Chart window with `unit_injector` model results, select **Page | Make** to get updated calculation results in the existed graphs (refer to *Figure 6-13* and *Figure 6-14*).

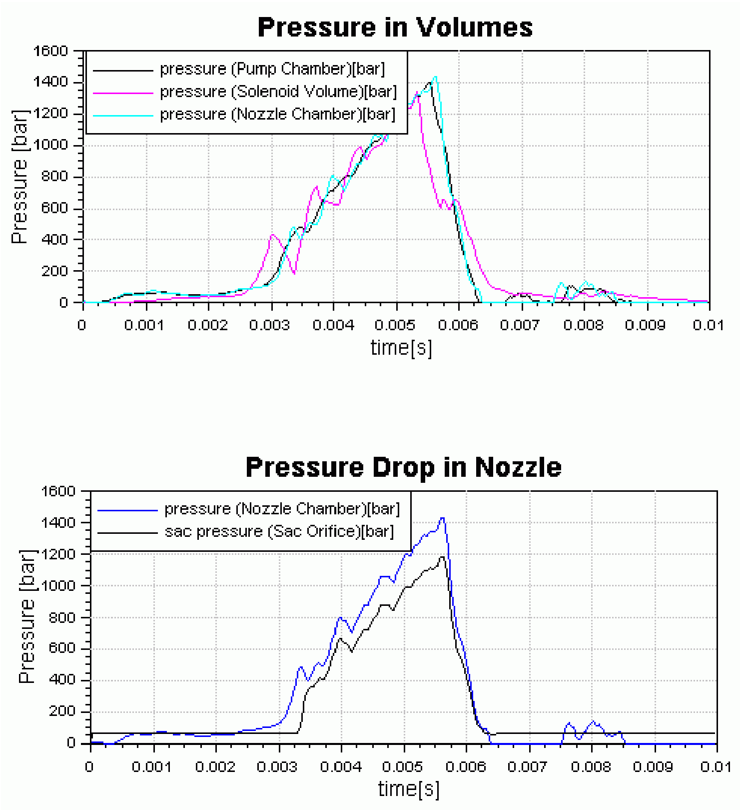


Figure 6-13: Pressures in Volumes (Standard Run + Restart)

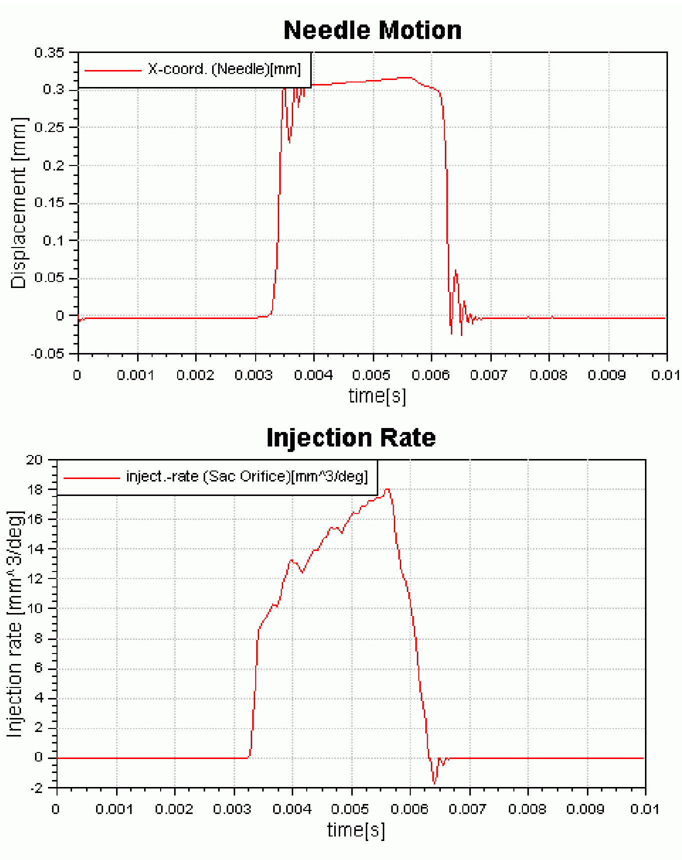


Figure 6-14: Needle Lift and Injection Rate (Standard Run + Restart)

The **Restart** option just appends new calculation results onto the previous results, i.e. you get a GIDas file with entire results (from the start point of the previous simulation to the end point of the restarted simulation).

6.1.7. Unit Injector with a Rocker Arm

The previous model of Unit injector comprised a Cam Profile element, which is defined with Follower acceleration at 1000 rpm (refer to *Figure 6-3*). In a real model, the unit injector is activated by a rocker arm (refer to *Figure 6-15*), so it is more convenient to replace the Cam Profile with a Rocker Arm (Cam) element from the Lever group.

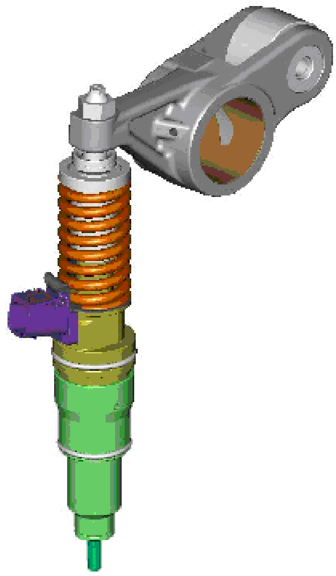


Figure 6-15: Unit Injector with Rocker Arm

Updated model of Unit injector is shown in *Figure 6-16*.

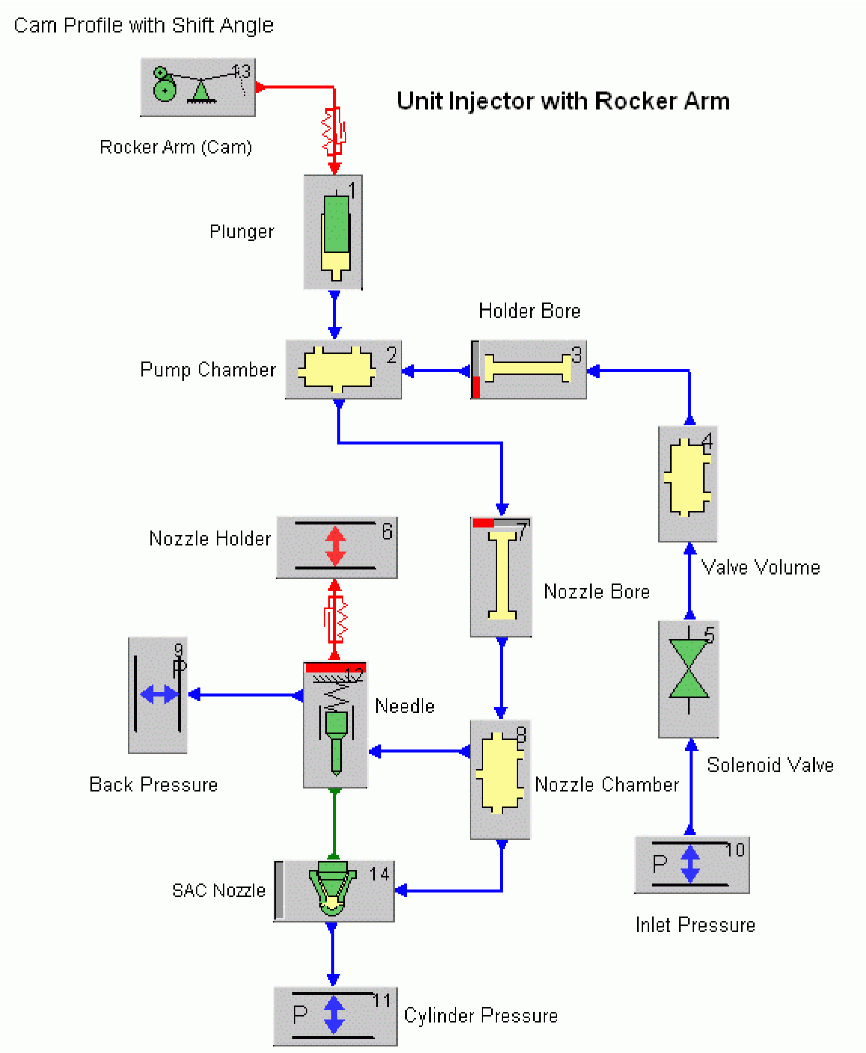


Figure 6-16: Model of Unit Injector with Rocker Arm

6.1.7.1. Rocker Arm (Cam)

The input dialog of the Rocker Arm (Cam) element is shown as follows:

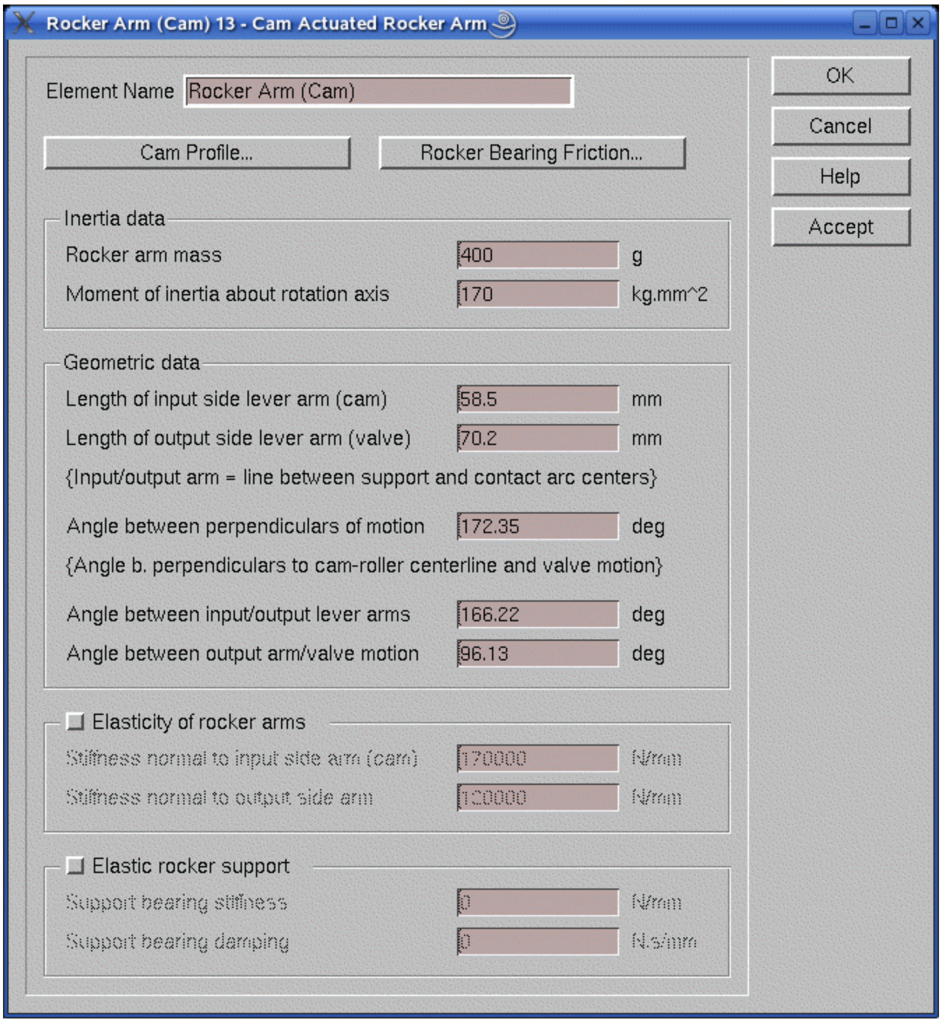


Figure 6-17: Input Dialog of Rocker Arm (Cam)

The following sub-dialog boxes are available and that it is not necessary to select a cam element separately for the cam and roller follower inputs:

- **Cam Profile**, into which cam and roller features must be input, including the direction of rotation of the cam.
- **Rocker Bearing Friction**, into which rocker bearing and frictional inputs can be input.

The Cam Profile sub dialog is shown in Figure 6-18. It must be activated and filled for any cam-operated system to be effective.

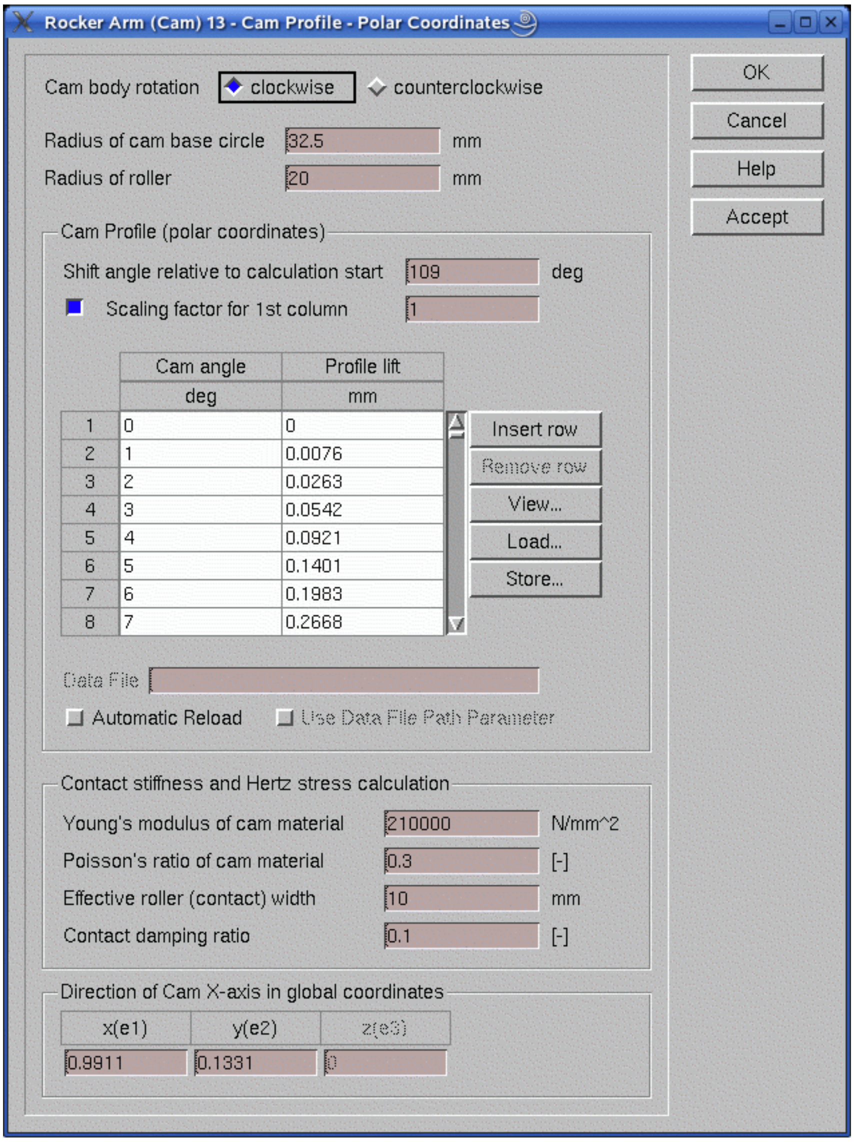


Figure 6-18: Input Dialog of Cam Profile / Rocker Arm (Cam)

In the previous example of Unit Injector, input data of Cam Profile was the Follower accelerations at 1000 rpm, which are measured. In this example, the real data of Cam Profile is used and its start is shifted for 109 deg from the calculation start.

The cam/roller contact stress is calculated by inputting the Young's modulus and Poisson's ratio for the cam material and the effective contact width of the roller. It is necessary for the calculation of rocker arm and cam shaft dynamic.

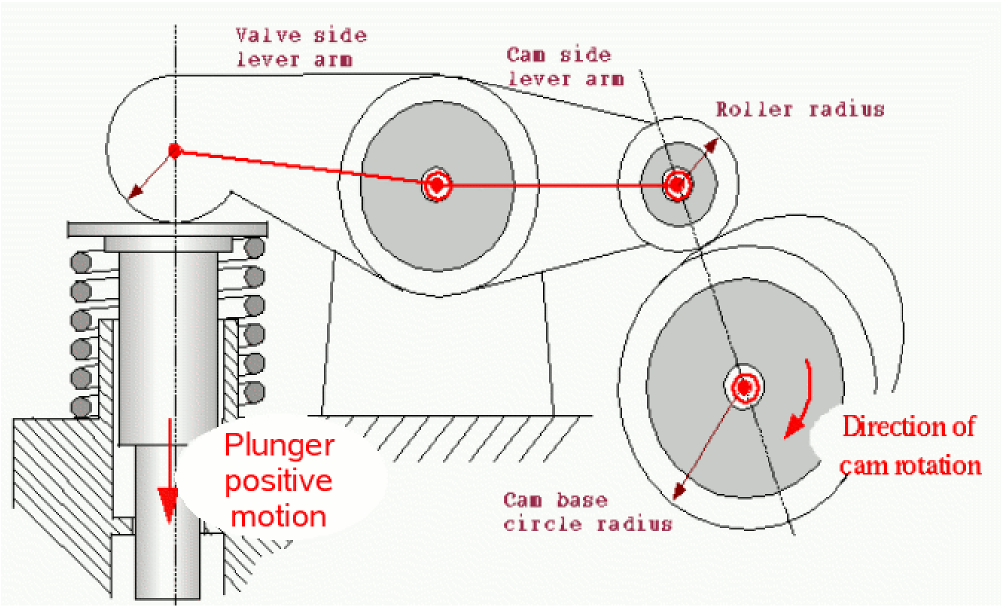


Figure 6-19: Rocker Arm (Cam) Schematic

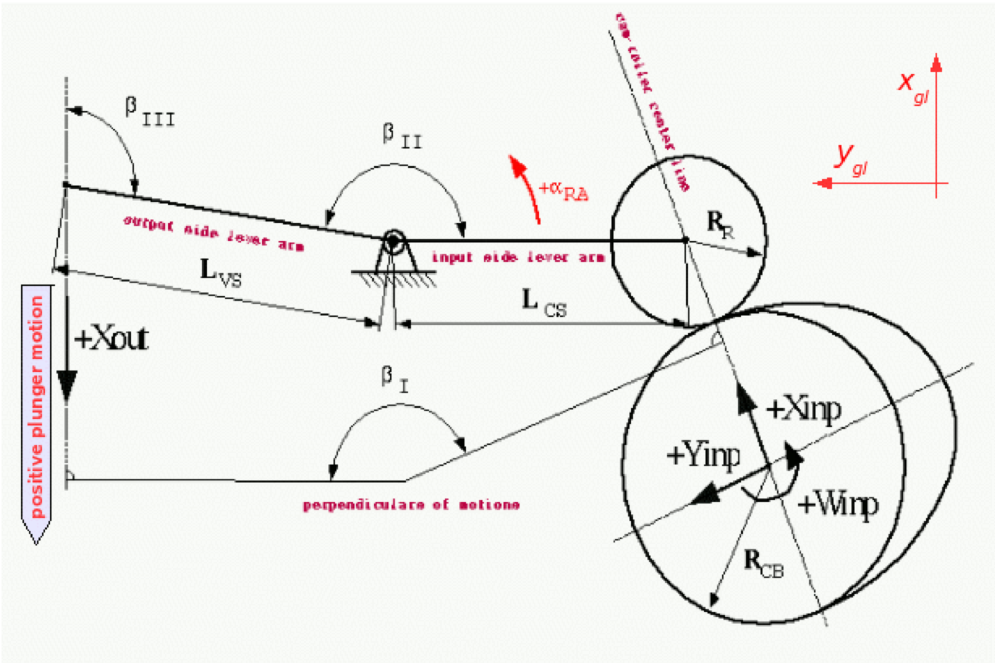


Figure 6-20: Rocker Arm (Cam) Geometry

A description of the features in this Rocker Arm (Cam) dialog is given below:

- The total mass of the rocker assembly (arm, roller, roller bush) and its moment of inertia around the rocker arm center of rotation axis (rocker support)
- Length of cam side lever arm (support-roller). This is the length of the arm from the rocker support axis to the roller center. Length LCS in Figure 6-20.
- Length of valve side lever arm (support-finger). This is the length of the arm from the rocker support axis to the center of curvature of the finger, which pushes the valve stem. Length LVS in Figure 6-20.

- Angle between perpendiculars of motion. This is the angle included between a line drawn perpendicular to the line joining the cam and roller centers and a line drawn perpendicular to the center-line through the engine valve. This is angle β I in Figure 6-20 and here it is a positive (see conventions in the Users Guide Chapter 5.2.1).
- Angle between cam and valve side lever arms. This is the angle included between the line drawn between the rocker support axis and the roller center (cam side lever line) and the line drawn between the rocker support axis and the center of curvature of the valve finger radius (valve side lever line). This is angle β II in Figure 6-20 and here it is a negative (see conventions in the Users Guide Chapter 5.2.1).
- Angle between output arm/valve motion. This is the angle between the line joining the center of curvature of the finger and the rocker support axis and the negative axis of the engine valve. This is angle β III in Figure 6-20 and here it is a positive (see conventions in the Users Guide Chapter 5.2.1).

Cam rotational direction

The direction of rotation of the cam has an effect on the kinematic motion of the roller, as the cam center is offset to the roller vertically.. This is accounted for in the model by specifying the direction of cam rotation and activating one of the toggle buttons in the Cam Profile sub-dialog.

6.1.7.2. Plunger (Direction of Element Motion)

The input dialog of the Plunger element is shown in the Figure below.

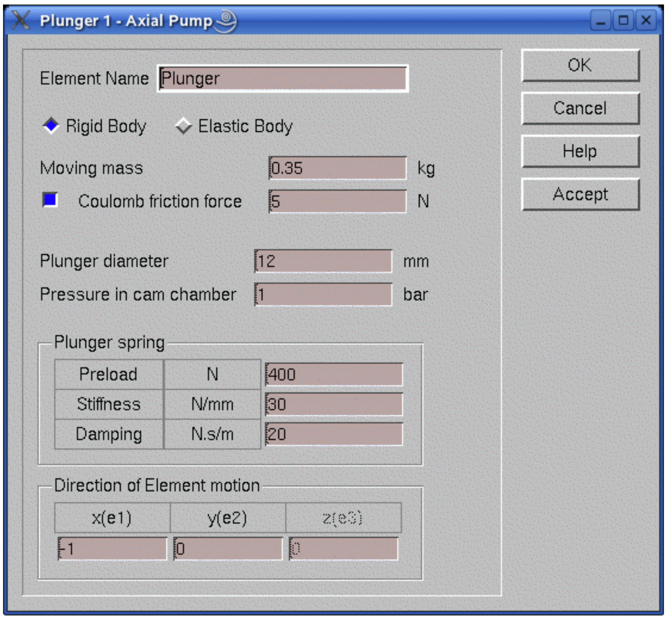


Figure 6-21: Plunger Input Dialog

In this dialog, the x direction of Plunger motion is set to -1 because the Plunger motion is opposite to the x-direction of the global coordinate system as shown in Figure 6-20 (positive plunger movement in its local coordinate system is downwards).

6.1.8. Running the Calculations

Save this change under a different file name, e.g. `unit_injector_ rockerarm.hyd`, and restart the calculation. Refer to *Sections 4.4* and *4.5* for running calculation and inputting results data in the Impress Chart window.

6.1.9. Calculation Results

Output parameters, which are plotted, are chosen the same as in Unit injector example (refer to *Figure 6-13* and *Figure 6-14*).

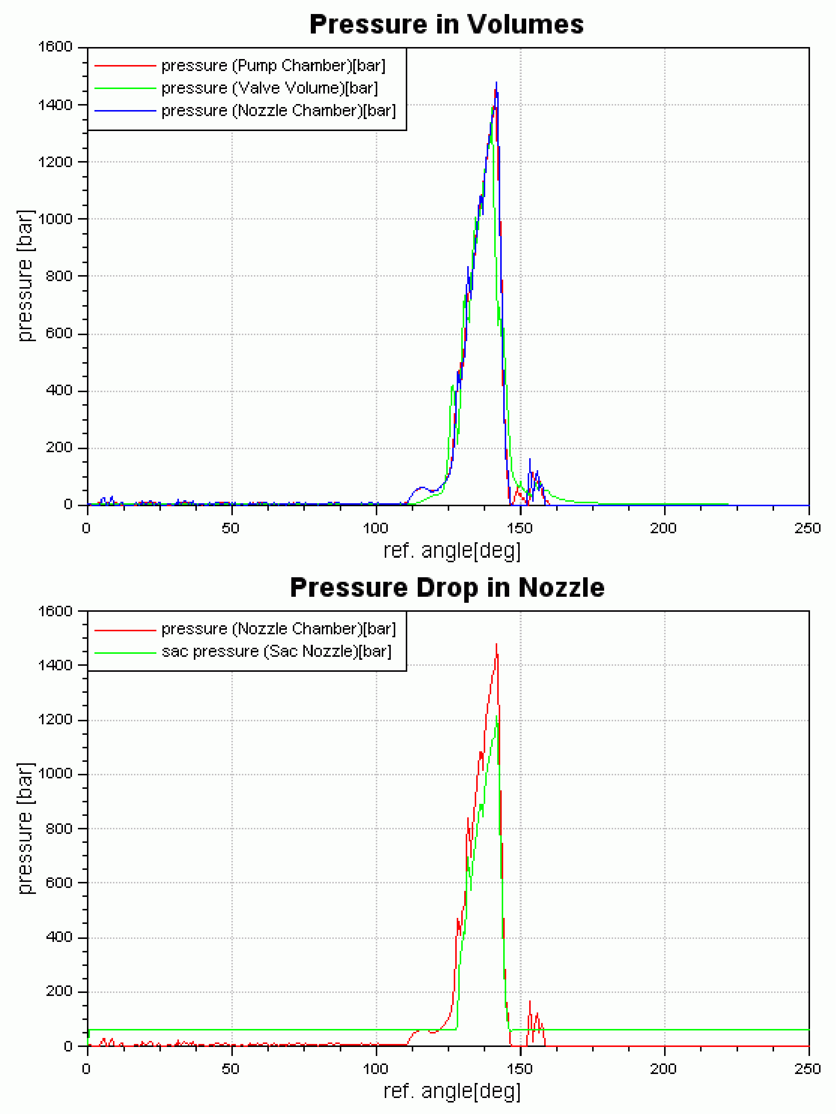


Figure 6-22: Pressures in Volumes

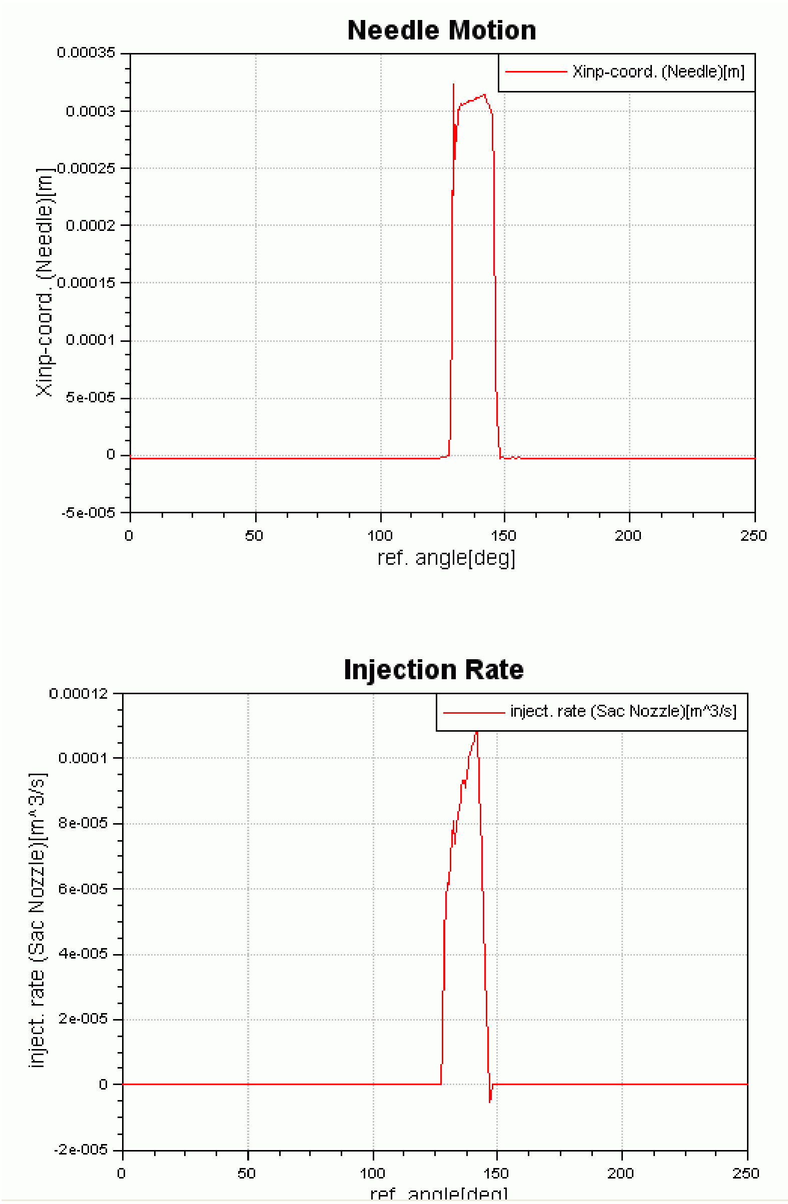


Figure 6-23: Needle Lift and Injection Rate

6.2. In-line Pump Injector

In this example the modeling and simulation of a conventional in-line pump (pump-line-nozzle) injection system as well as Search Adjust (1D optimization) and Animation of nozzle flow, will be demonstrated. The system is outlined in *Figure 6-24*. It consists of the in-line pump, constant pressure delivery valve, injection line and injection nozzle. The BOOST Hydsim model of the system is shown in *Figure 6-25*. Notice that, instead of a constant pressure delivery valve, other different valves can be used in the in-line pump injection systems: snubber valves, constant volume retraction valves (with and without back flow orifice), etc.

As in the previous example, the in-line pump is driven by Cam Profile 1, which is a mechanical boundary defining kinematics excitation (acceleration in this case). The cam profile itself is taken to be rigid. The stiffness and damping of the entire drive (e.g. camshaft) is represented by the mechanical connection between Cam Profile 1 and Plunger 2. In many cases, this simplified representation of the drive stiffness provides a sufficient accuracy.

The in-line pump itself consists of Plunger 2, Round Spill Port 3, and Pump Chamber 4. Plunger 2 is different from the plunger in the unit injector example. It has a complex geometry with a side groove (or central bore) and a helix slot (metering ramp) for fuel spilling (for a detailed description refer to the Users Guide). This plunger must be connected to one or several spill ports for fuel spilling into the Feed Pressure 15. The Feed Pressure 15 is modeled as a hydraulic boundary condition with constant pressure of 3 bar. If spill ports have equal geometry (diameter and height in the pumping barrel), they can be modeled by one spill port element only, otherwise each port has to be represented by a separate spill port icon.

Delivery Valve 5 is located between the pump and the injector. It supplies the fuel in the delivery direction: from Pump Chamber 4 into the Valve Chamber 6 and further to the Injection Line 7. To model a constant pressure valve in BOOST Hydsim, a Check Valve 14 must be switched in parallel with the Delivery Valve 5 as shown in *Figure 6-25*. Check Valve 13 must act in the opposite direction, i.e. it must have opposite connections to Pump Chamber 4 and Valve Chamber 6 than Delivery Valve 5. Thus, Check Valve 13 allows only the backward flow from Valve Chamber 6 to Pump Chamber 4. Input connection always denotes the flow direction for Delivery and Check Valves. In this way, the hydraulic connections to these valves are irreversible.

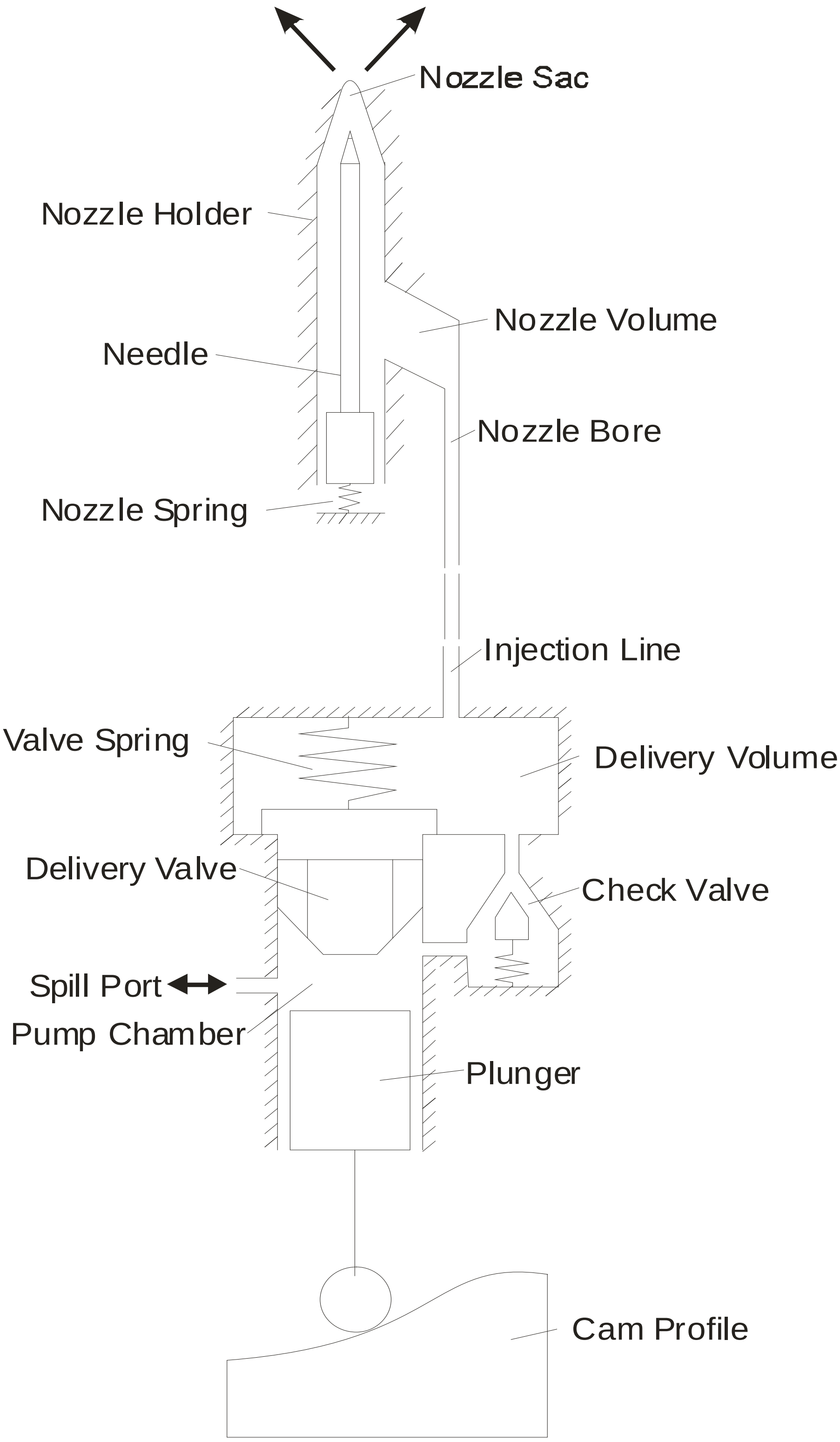


Figure 6-24: Schematic of an In-line Pump Injector with Constant Pressure Valve

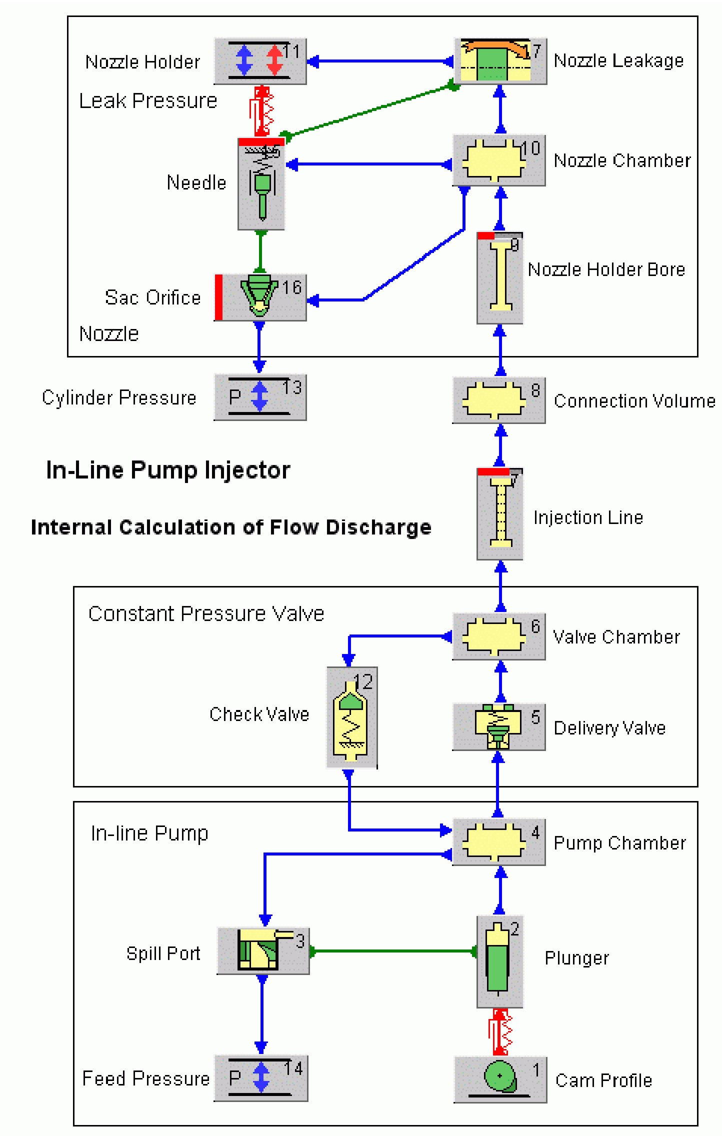


Figure 6-25: BOOST Hydsim Model of the In-line Pump Injector

Until now, most of the fuel lines in our examples were modeled by Laplace Line model because it is the fastest and thus most often used. However, in this example, the usage of a Characteristics Line for the Injection Line 7 is demonstrated. Its main advantage over a Laplace Line is that it calculates the pressure distribution along the line (at up to five locations). In this model, the pressure wave equation along the line is solved by the well-known characteristics method. Pressure output at different line cross-sections is often useful for performing the comparison between the measured and calculated pressures at the specified locations of the injection tube (on pump and nozzle side). Injection Line 7 is connected via a small Connection Volume 8 to the Nozzle Holder Bore 9 which is modeled as a Laplace Line element. It should be emphasized that Laplace and Characteristics Line models are completely compatible because they use the same model for the calculation of nonstationary frictional losses (for more information consult BOOST Hydsim Users Guide). Notice that in BOOST Hydsim the two different Line elements cannot be connected directly: a small transition volume has always to be inserted between them.

The injector comprises the following elements: Nozzle Holder Bore 9, Nozzle Chamber 10, Needle 16, Nozzle SAC Orifice 17 and Nozzle Holder 11. In this example, we have chosen an extended SAC nozzle orifice model with cavitation SAC(extended) to show BOOST Hydsim capabilities of modeling fuel flow through needle seat and nozzle spray holes. The leakage through the needle guide is modeled by Nozzle Leakage 12. This element is not of primary importance and in many cases may be omitted. Cylinder Pressure 14 represents the boundary pressure for fuel injection into the engine cylinder.

6.2.1. Creating the Model

The complete BOOST Hydsim model of the in-line pump injection system in *Figure 6-25* consists of the following elements:

- Cam Profile (**CAM/Cam Profile**)
- Plunger (**PUMP/Plunger**)
- Spill Port (**PORT/In-line Fill/Spill**)
- Pump Chamber (**VOLUME/Standard**)
- Delivery Valve (**VALVE/Delivery**)
- Valve Chamber (**VOLUME/ Standard**)
- Injector Line (**LINE/Characteristics**)
- Connection Volume (**VOLUME/ Standard**)
- Nozzle Holder Bore [**LINE/Laplace Model)**
- Nozzle Chamber (**VOLUME/ Standard**)
- Needle [**NEEDLE/Standard (up-to-date model)**]
- Nozzle SAC Orifice [**NOZZLE/SAC (extended)**]
- Nozzle Holder(Leak Pressure) (**BOUNDARY/Hydromechanical**)
- Nozzle Leakage (**PUMP/Leakage**)
- Check Valve [**VALVE/Check (poppet)]**
- Cylinder Pressure (**BOUNDARY/ Pressure**)
- Feed Pressure [**BOUNDARY/Pressure**)

Notice that Nozzle Holder and Back (Leak) Pressure are combined into one boundary condition element 11. This is possible because the mechanical and hydraulic boundary conditions are independent and can be specified within one element. The connection between Needle 16 and Nozzle Holder/Leak Pressure 11 is mechanical. However, in this particular case, it also transmits hydraulic pressure to the backside of Needle 16. In general, any mechanical connection to a Hydromechanical Boundary would also transmit the hydraulic pressure and flow (if defined in the boundary table) from the boundary to the connected element (if this element accepts pressure and/or flow).

The user may observe that there are several new elements in this model which were not used in the previous examples: Spill port 3, Delivery Valve 5, Check valve 13 and some others. Their input data are briefly described below.

6.2.1.1. Spill Port

The first new element is the round Spill Port 3. This element must always be connected (by a special connection – green line) to a plunger and must also have two hydraulic connections to volumes or hydraulic boundaries. The input dialog of the round Spill Port is as shown in *Figure 6-26*:

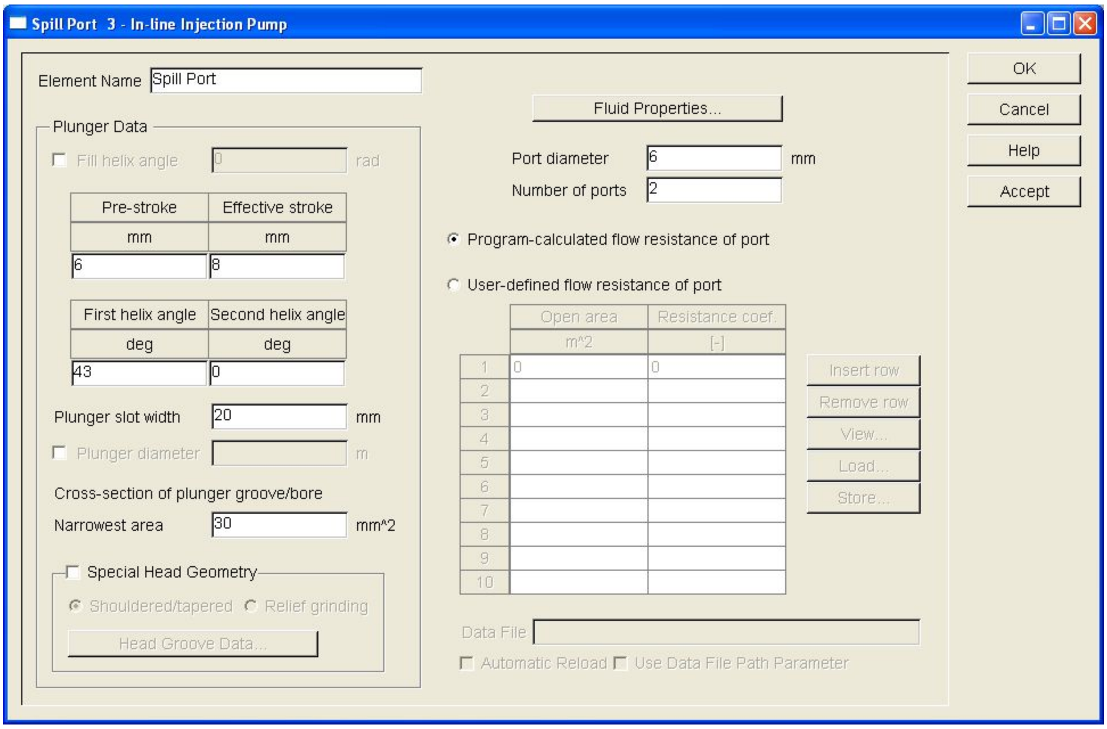


Figure 6-26: Input Dialog Box of Spill Port Element

In the above dialog, the user has to specify the following:

- Port diameter
- Number of ports
- Flow resistance coefficient at the narrowest cross-section of the port (constant or as a function of cross-sectional flow area). If the program-calculated flow resistance coefficient button is activated, it is calculated internally as a function of the narrowest cross-sectional area of the port. For a detailed explanation of spill port and plunger models, refer to the BOOST Hydsim Users Guide.
- Plunger data: pre-stroke, effective stroke, helix angle(s), width of plunger slot and narrowest cross-sectional area of plunger groove (or bore). In BOOST Hydsim, there exists also another spill port element: Distributor Spill of a distributor-type injection pump. It is a rectangular spill port exclusively used with the Cam Plate element from CAM group (not with Cam Profile). The user must be careful to select the right element and not mix the two spill ports with each other.

- If several load points are to be run, using Case Explorer within the same model file, different plunger Pre-stroke values may be required (to achieve the required different start of injection timings) and different Effective strokes will certainly be required (for different fueling quantities). The required injection timing changes (pre-strokes) can normally be assessed from knowledge of the total system length and the natural injection advance of the system as cam speed reduces. These can be reasonably accurately accessed using the following empirical formula:

$$\alpha_{inj} = 4.5 L_{line} \frac{\Delta n}{1000},$$

- where α_{inj} is the injection advance/delay in degrees of cam angle, L_{line} is the total line (system) length in meters from the top of delivery valve assembly to the nozzle seat and Δn is the change of cam (reference) speed in rpm.
- Hence, for example, if engine speed drops from rated speed of 1500 rpm to idle speed of 400 rpm (cam speed drops correspondingly from 750 rpm to 200 rpm) and the line length from delivery valve to nozzle seat is 1.10 m, the natural (dynamic) injection advance as speed drops would be:

$$\alpha_{inj} = 4.5 \times 1.10 \frac{1500 - 400}{2 \times 1000} = 2.7 \text{ deg cam}.$$

- This is often useful if special fuel pump timing helices have to be developed for variable speed engine applications or for approximate calculation of initial timing requirements for electronic pump-line-nozzle systems, before detail testing in hardware.
- The **static effective stroke** of the plunger (stroke without pump pressure) for each load case can be calculated by dividing the required fueling quantity through the plunger area:

$$x_{eff\_st} = \frac{Q_{inj}}{A_{pl}},$$

- where Q_{inj} is the required fueling quantity and A_{pl} is the plunger cross-sectional area.
- If a Constant Volume Retraction valve is used, its unloading volume has to be taken into account. In this case the following formula should be used:

$$x_{eff\_st} = \frac{Q_{inj} + V_{unl}}{A_{pl}},$$

- where V_{unl} is the unloading volume.
- BOOST Hydsim calculation actually requires a **dynamic effective stroke** of the plunger on input. Without considering the pump and nozzle leakage and mechanical deformation of the camshaft-plunger assembly, this **dynamic effective stroke** can be expressed by the following equation:

$$x_{eff\_dyn} = \frac{Q_{inj}}{A_{pl}} + \frac{\Delta p V_{\Sigma}}{E A_{pl}},$$

- where Δp is the pressure rise during injection, E is bulk modulus of the fuel and V_Σ is the total volume in the pump chamber, injection line and nozzle. The second term accounts for the compressibility of the fuel. The calculated effective stroke value is anyway not exact because the pump and nozzle pressure varies during injection and there is a time delay between them.
- Later, in *Section 6.2.5* it will be shown how the BOOST Hydsim program can be used to iterate the calculation automatically to reach the required injection timing or fueling by selecting the required parameters for it. In this case, the **static effective stroke** can be used as a starting value for the BOOST Hydsim calculation.

6.2.1.2. Delivery Valve

The next new element is the Delivery Valve 5. As stated earlier, it connects Pump Chamber 4 to Valve Chamber 6 and allows flow only in the delivery direction (irreversible connection). The input dialog of this element is as follows:

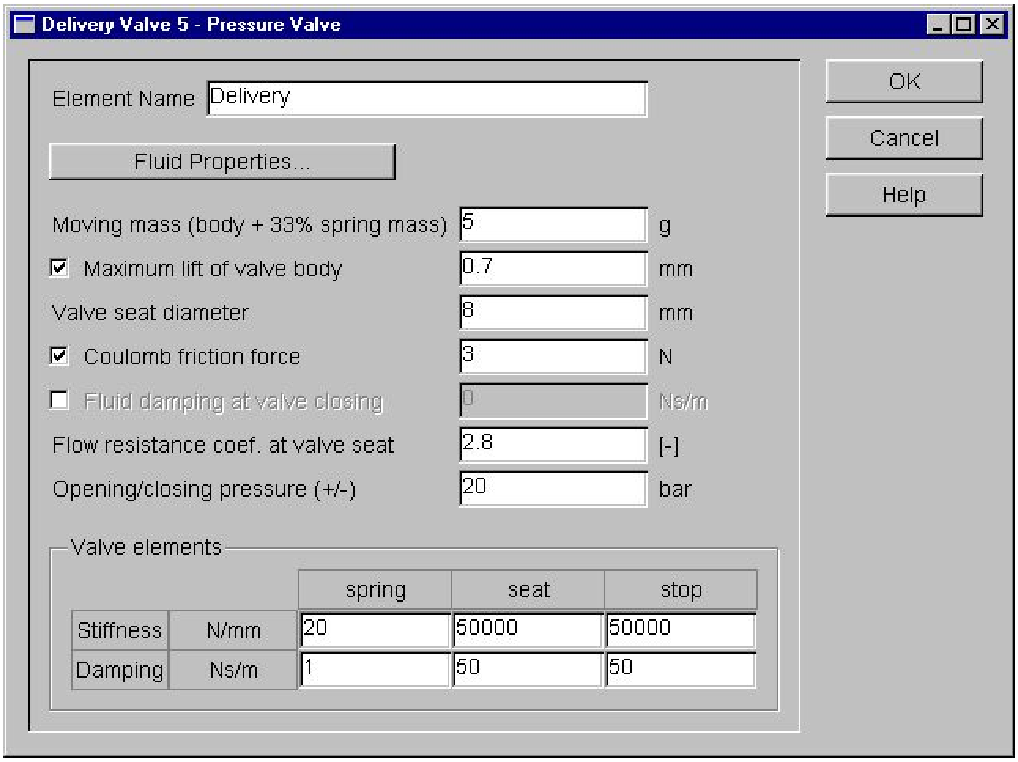


Figure 6-27: Input Dialog of Delivery Valve

Here the user must specify a number of valve parameters such as:

- Moving mass (valve body mass + 33% spring mass)
- Valve seat diameter
- Opening pressure
- Flow resistance coefficient at fully open valve
- Armature lift, etc.

Stiffness and damping rate of valve spring, seat and stop must also be entered in the input dialog. For valve seat and stop the data need not be specified precisely: their stiffness must only be high enough not to interfere with the operating natural frequencies of the system. Damping values of seat and stop are usually set based on experience.

If a snubber valve is used instead of a constant pressure valve within a pump-line-nozzle injector, the Check Valve in *Figure 6-25* has to be substituted by an Orifice element. The user may try it as an exercise. As stated earlier, other delivery valve types (e.g. Constant Volume Retraction valve) can be modeled as well.

6.2.1.3. Check Valve

Another new element in our system is Check Valve 13. In BOOST Hydsim, there are two types of pressure check valves: valve with spherical body Check (Ball) and valve with conical body Check (Poppet). In this example, the valve with conical body is used. Its input dialog is shown in *Figure 6-28*:

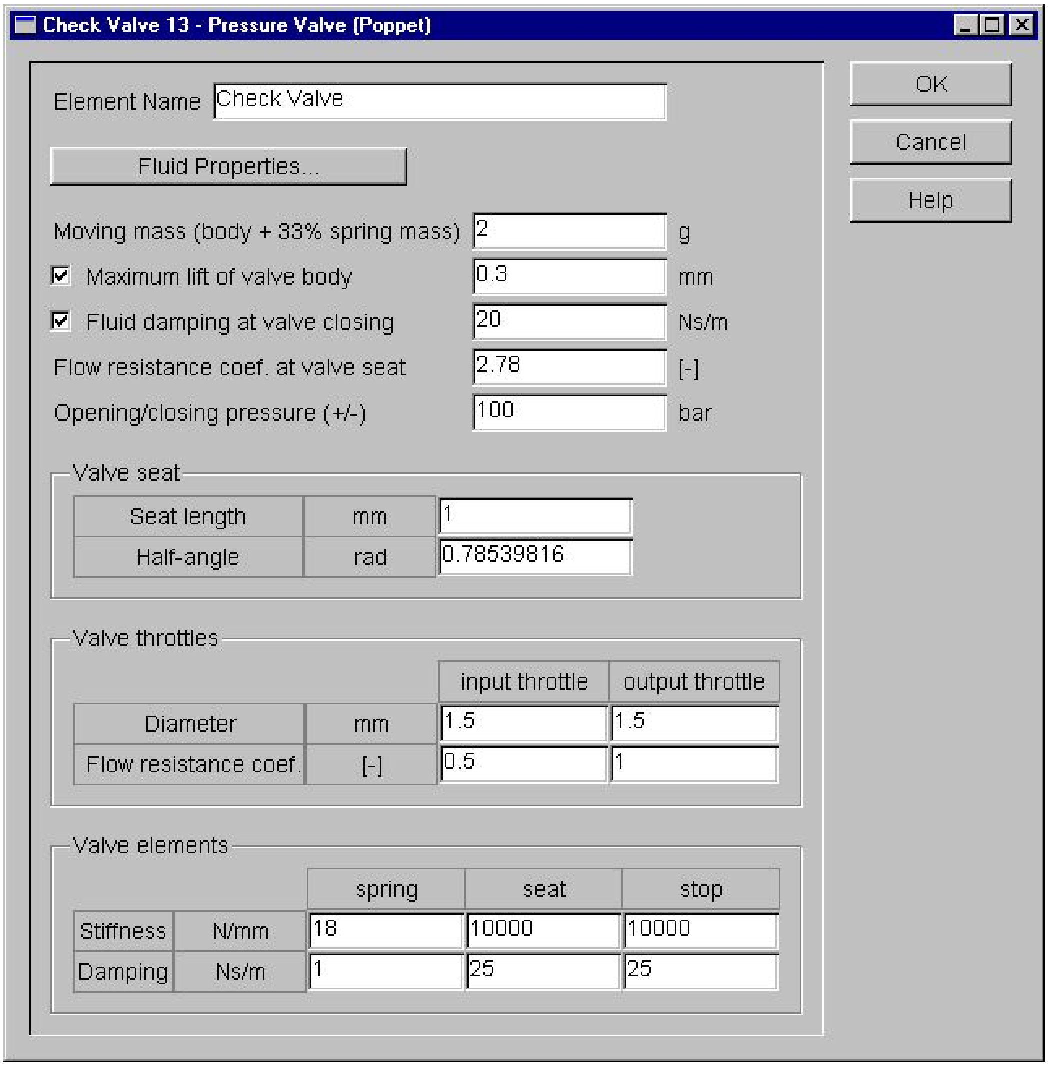


Figure 6-28: Input Dialog of Return-Flow (Check) Valve (Poppet)

The input data dialog of the pressure check valve has much in common with the input dialog of the delivery valve. Additionally, the geometry of the conical valve seat and the data for the input/output throttles has to be specified. Default value of the flow resistance coefficients at the input throttle is 0.5 (expansion of the cross-sectional area) and at the output throttle - 1 (contraction of the cross-sectional area).

Check Valve is used to keep a constant pressure in the Injection Line, thus preventing cavitation effects and hydraulic instability. They are often used in variable speed fuel injection applications where the retention of a good 'residual' pressure in the line between pump and nozzle is very beneficial at the lower load/speed and idling conditions, for improved fuel atomization and injection shot/shot consistency, hence improved engine combustion efficiency.

Valve Check (Ball) has a ball as the valve body and thus does not require seat length as input data. Otherwise it is analogous to the check valve with conical body (Poppet).

6.2.1.4. Injection Line

The next important element of our injection system is Injector Line 7. As stated earlier, it is modeled by a **Characteristics Line** element, which solves the wave equation by the classical method of characteristics. This method is numerically involved and may result in relatively long computational time compared to the semi-analytical **Laplace Line** model. The input data dialog of this line element has the following form:

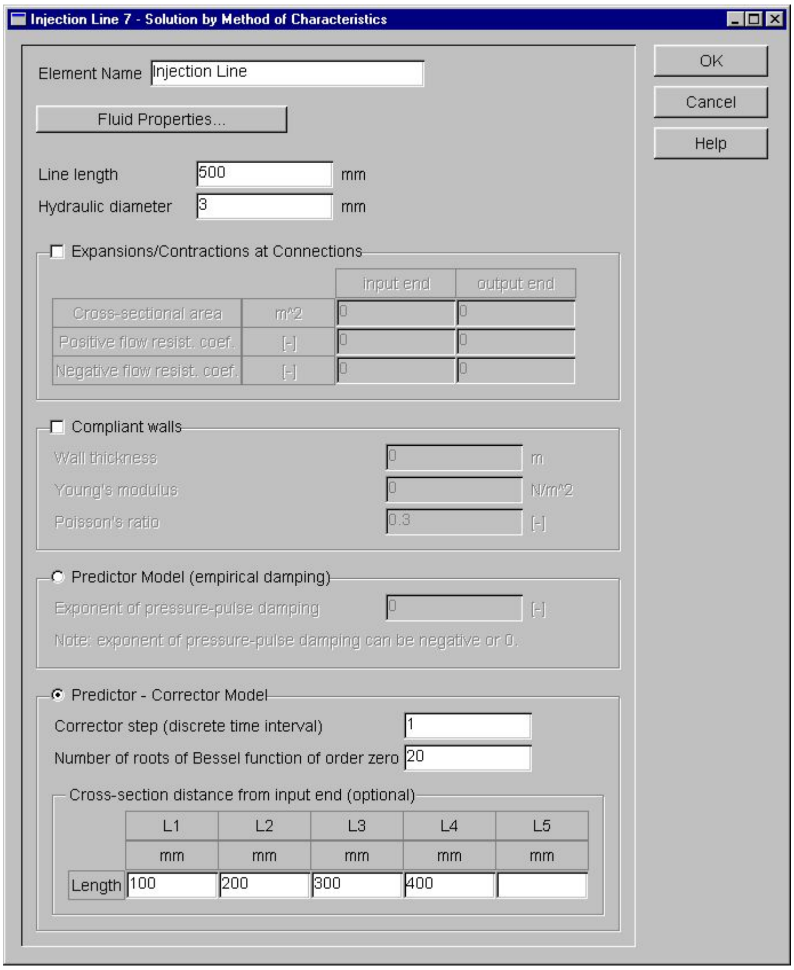


Figure 6-29: Input Dialog of Injection Line (Characteristics model)

The Characteristics Line may have contractions and/or expansions at the line ends. If the button Expansions/Contractions at Connections is activated, the respective cross-sectional areas and flow resistances have to be specified in the input dialog. Note that in case a non-zero area at input connection is specified, the flow resistances in the positive flow direction (from input end towards tube) and in the negative direction (from tube towards input end) must be defined. Default values for flow resistances are 0.5 and 1 respectively. Analogously, if a non-zero area at output connection is entered, the flow resistances in the positive flow direction (from tube towards output end) and in the negative direction (from output end towards tube) must be specified. Default values are 1 and 0.5.

The discrete method of characteristics is used for the solution of the line equation only if the Predictor-Corrector Model is chosen as shown in *Figure 6-29*. In this case the corrector step (positive integer value) has to be specified. If it is set to 1 (default), then each time step for the “discrete characteristics model” is a corrector step, i.e. the non-stationary damping function is recalculated at each time step. Thus, we have a pure corrector model. This model, however, may lead to a large number of discrete line section (cells) and cause long computation time because the numerical integration of the in-line pump and other injection systems requires a small time step. To reduce the computational effort, the user should set the corrector step to a higher integer value (2,3,4,N). This would imply every N-th step as a corrector step and the steps in between as predictor steps. At each corrector step the damping function is calculated for the predictor model (for details refer to the BOOST Hydsim Users Guide).

If the Predictor-Corrector Model is used, the user may select up to five line cross-sections (L1...L5) for the output. Each cross-section is defined by specifying its distance from the line input end.

If the user wishes to define his own (empirical) damping function of the pressure wave in the line, he has to use another Predictor Model option within the input dialog. In this case the pressure-pulse damping (negative value) must be specified. The characteristics for this model are calculated only at the line ends, therefore no output in other line cross-sections is possible. The user should try this option and see that, as soon as the button Predictor Model (empirical damping) is activated, the field of cross-sections L1...L5 is grayed out, i.e. no data can be entered there. In fact, the Predictor Model is nothing else than a well-known d’Alembert solution of the wave equation with an empirical loss function. The same model is available in the d’Alembert Line model, which is also accessible from LINE group. If the pressure-pulse damping is set to 0, no frictional losses would be considered and the model will yield an exact d’Alembert solution of the wave equation. Because of the empirical loss function, the pure Predictor Model should be used only in special cases.

If the change of sound velocity in the line due to wall compliance has to be modeled, the user has to click the field Compliant walls and specify wall thickness, Young’s modulus, and Poisson’s ratio for the line material. Note that the distention of the line walls is not explicitly considered in the present Characteristics Line model (refer to the BOOST Hydsim Users Guide for more information). Of course, output pressure is available only at those cross-sectional areas which are defined in the output dialog box (*Figure 6-30*), from 1 to 4 in this example, at 100 mm steps in the 500 mm long line.

As output parameters for any line type, the user could choose pressures, volumetric and cumulative flow rates at the line input and output ends (in Time and Reference angle domains if he/she wishes). For the Injection Line of our in-line pump model, we select pressures at both the input and output ends and four cross-sections, as shown in *Figure 6-30*:

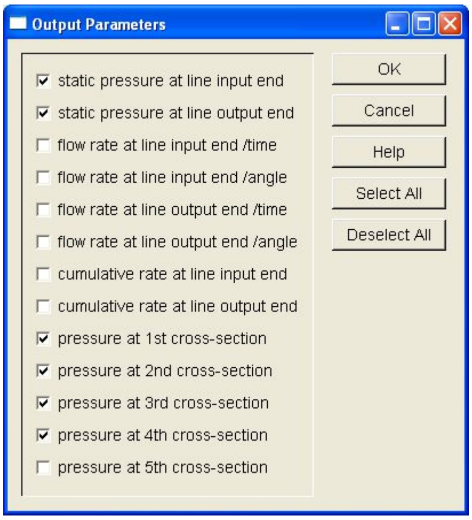


Figure 6-30: Output Data of Characteristics Line

6.2.1.5. Needle

The next element is the Needle 16. Its Input dialog box is already shown in Figure 5-10. Here we will discuss the Needle seat/stop stiffness and damping. In Figure 6-31 the input dialog box of standard Needle (up-to-data model) is shown:

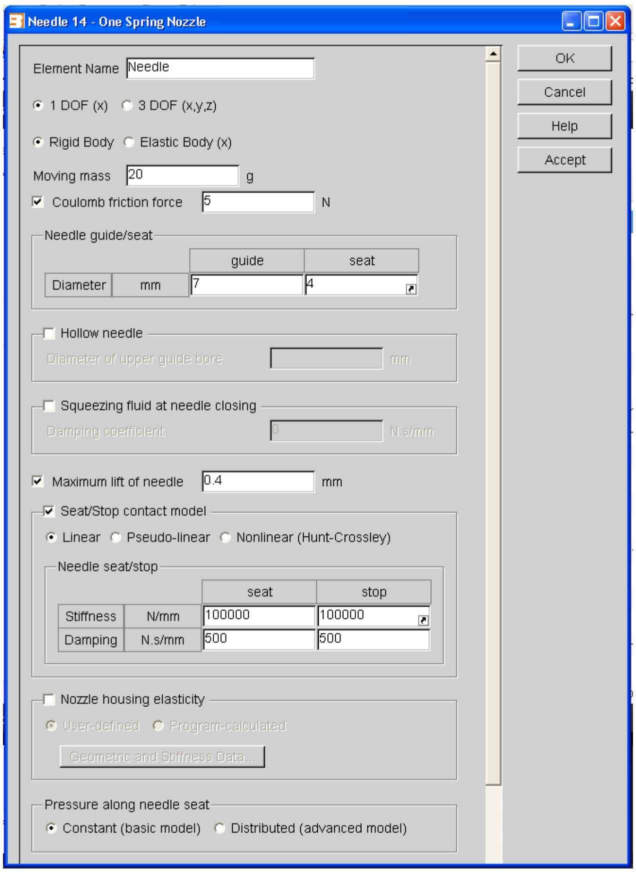


Figure 6-31: Input Data Dialog box of Standard Needle (up-to-date model)

Needle seat/stop stiffness is 100000 N/mm, and damping is taken as 500 Ns/m, which is 15-20% of the critical damping. Critical damping (from the vibration theory) is calculated by:

$$C_{crit} = 2 \cdot \sqrt{mass \cdot stiffness}\,. \qquad (6.2.1)$$

6.2.1.6. SAC Nozzle Orifice

The last new element of our model is Nozzle SAC Orifice 17. It is an Extended SAC Orifice model with cavitating flow (icon SAC/Extended model from NOZZLE group). The user is already familiar with the basic SAC Nozzle Orifice model from the common rail example with three-way solenoid valve (refer to *Section 5.2*). The input data for the extended nozzle orifice model is partially the same but a number of new parameters are required as shown in the dialog that follows:

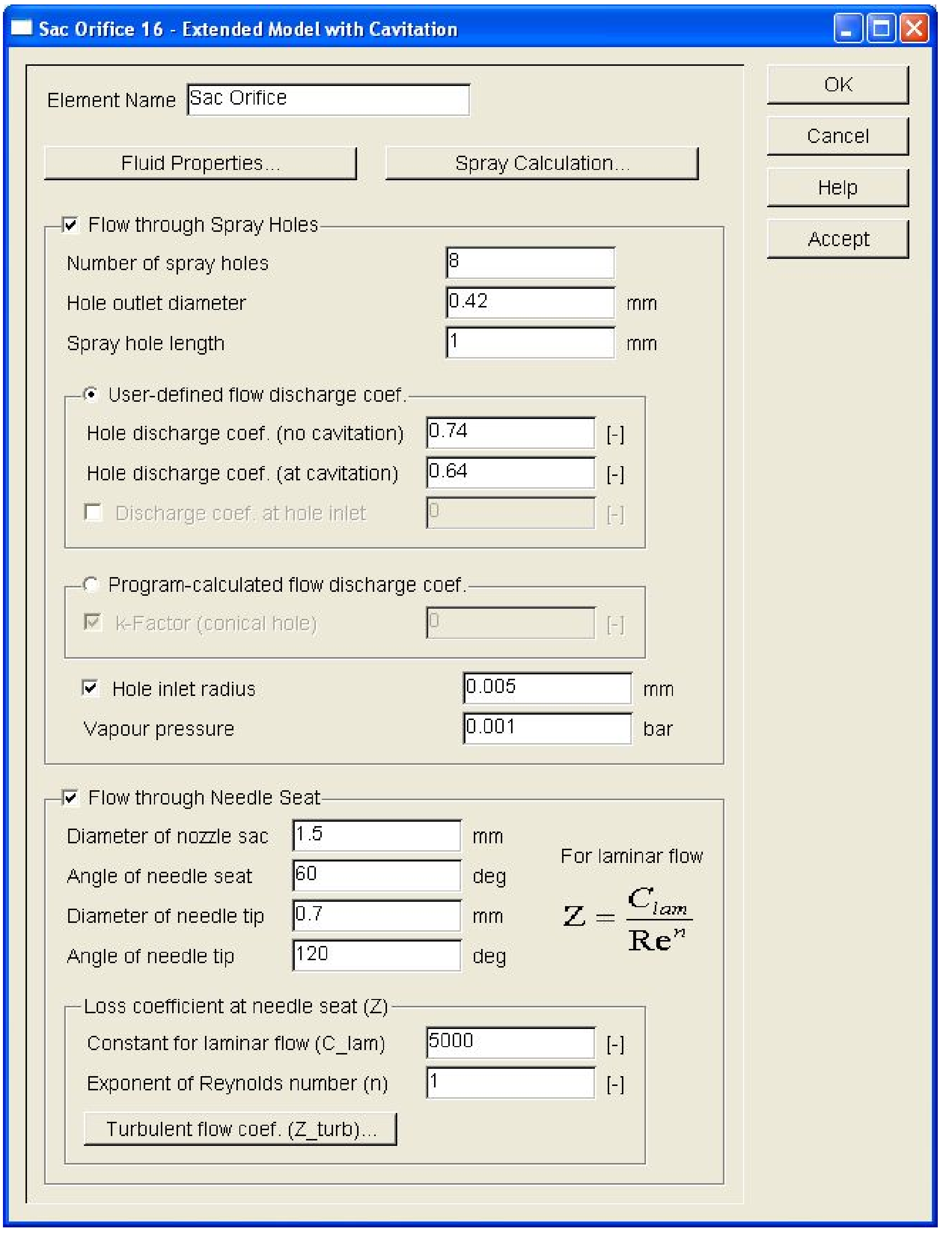


Figure 6-32: Input Dialog of Extended SAC Nozzle (with User-defined Flow Discharge)

In the above dialog, the user has to additionally specify the diameter and the included angle of the needle tip and two flow discharge coefficients at the nozzle spray holes: one for hydraulic flow and one for cavitating flow. The flow coefficient at cavitation can be estimated only from experimental data. Furthermore, the user has to enter the parameters for calculation of the loss coefficient at the needle seat for laminar flow. BOOST Hydsim uses the following formula for this

$$Z = \frac{C_{lam}}{Re^{n}}, \tag{6.2.2}$$

where C_{lam} is the numerator constant, Re is the Reynolds number and n is its exponent. AVL default estimates are 3500 for C_{lam} and 1 for n. *Formula 6.2.2* with the exponent of Reynolds number in the denominator allows a higher flexibility in modeling the laminar flow at needle opening in comparison with classical formula of this type.

The BOOST Hydsim cavitation model does not attempt a rigorous physical modeling of a two-phase flow for nozzle spray holes but substitutes it by an equivalent model with another discharge coefficient. This approach based on a modified Bernoulli equation, shows satisfactory results in many applications. Notice that at cavitating flow, only the input pressure (pressure in the nozzle sac) is taken for calculation.

The flow resistance coefficient of the needle seat at turbulent flow is set to 1 by default. It may be defined also as function of needle lift at opening and at closing procedure. Turbulent flow occurs only at higher needle lifts, for which the resistance at the needle seat is assumed to be negligible, compared to the flow resistance at the nozzle spray holes. At low needle lifts, flow through the seat is laminar and throttling both at the needle seat and at the spray holes is considered. At higher needle lifts (turbulent flow through the seat), throttling is considered only at the nozzle spray holes (refer to the BOOST Hydsim Users Guide for more information). In fact, the resistance coefficient at the spray holes may indirectly include the flow discharge effects at the needle seat too. The user is free to specify any other relevant flow resistance coefficients if they can be verified by experimental data.

In this model, the flow discharge coefficient in spray hole is provided directly on input. To do this, the option **User-defined flow discharge coef.** has to be activated. Two **hole discharge coefficients** (at hydraulic and cavitating flow) have to be specified there. **Discharge coefficient at hole inlet** is an optional parameter. It can be defined for the considering of additional losses at hole inlet, if these are not already encountered in the previous discharge coefficients.

Save this model under name `inline_pump.hyd`.

An alternative possibility is the internal BOOST Hydsim algorithm for the flow discharge calculation (refer to BOOST Hydsim Users Guide). To apply it, the option **Program-calculated flow discharge coef.** has to be activated (refer to *Figure 6-33*). In this case, the hole discharge coefficient is calculated internally during the entire injection event. This model takes into account the nozzle spray hole configuration, pressure distribution in the spray hole, flow losses and cavitation formation. Here only two input parameters are required: **hole inlet radius** (optional) and **vapour pressure**.

Angle of needle tip I the angle of the needle tip cone.

Diameter of needle tip is the base diameter of needle tip cone.

Save model under a different name, for instance `inline_pump1.hyd`.

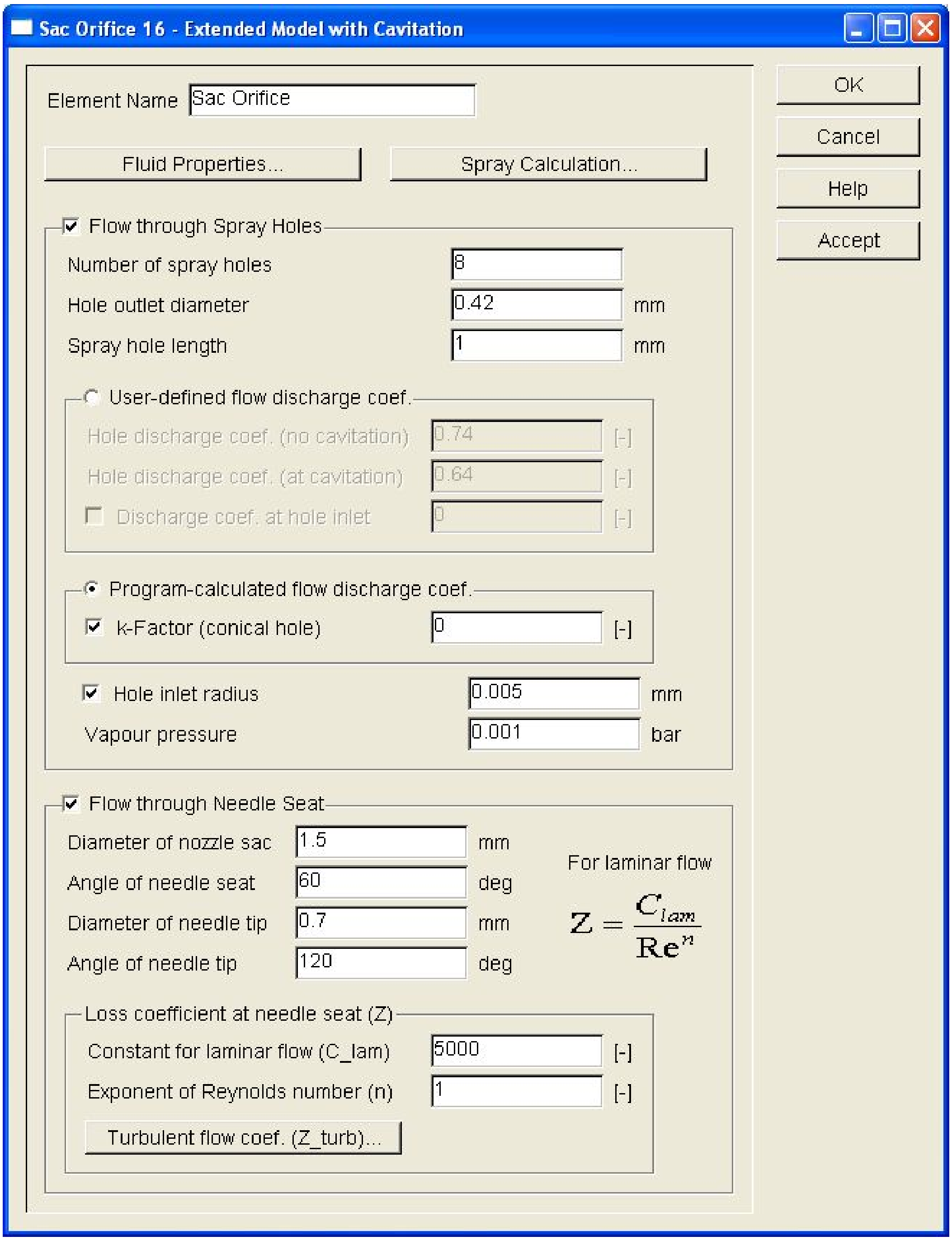


Figure 6-33: Input Dialog of Extended SAC Nozzle (with Program-calculated Flow Discharge)

BOOST Hydsim model for the calculation of hole discharge coefficient takes into account the nozzle hole geometry and flow characteristics at hole inlet, vena contracta and hole exit. Therefore the variation of the inlet radius and hole length has significant influence on the hole discharge coefficient. For detailed information refer to BOOST Hydsim Users Guide.

6.2.2. Running the Calculations

Before starting the calculation, the user must specify the initial pressure in volumes (from menu **Element | Initial Values**), calculation control data (from **Simulation | Control**) and fluid properties (from **Simulation | Fluid Properties**). Pump Chamber should have a low initial pressure (equal to Feed Pressure) while Nozzle Chamber, Connection Volume and Valve Chamber should have high initial pressures, equal to that pressure trapped in the injection line by the delivery valve after injection. Furthermore, desired output data in **Element | Store Results** must be defined.

For performing the calculation, refer to *Section 4.4*.

6.2.3. Calculation Results

Important calculation results for a pump-line-nozzle system are pressures in volumes (Pump Chamber, Valve Chamber, Connection Volume and Nozzle Chamber) and along the Injection Line. These pressures are plotted in *Figure 6-34* (at a camshaft speed of 500 rpm). Obviously, all pressure curves have similar character and, in this case, maximum values of around 1200-1300 bar. Pressure in the Pump Chamber has a steep drop at 37 degrees (at this instant the Plunger helix reaches the Spill port and the fuel starts spilling into the pump gallery). Note that sac pressure in this particular case is considerably smaller than the nozzle pressure (maximum value 800 bar against 1300 bar). This is related to a high throttling at needle seat because we specified the value 1 for the **turbulent flow constant (Z_turb)** at needle seat in the input dialog of the SAC nozzle. Normally, at full needle lift sac and nozzle chamber pressures would not differ that much. If flow throttling at needle seat is neglected (i.e. **turbulent flow constant** at needle seat switched off or set to zero), the sac and nozzle chamber pressures will be practically equal. The user can try it as an exercise.

Other typical output parameters are Needle motion and Injection rate. These are shown in the top and middle graphs of *Figure 6-35*, respectively. You may observe secondary injection, which is the result of a returned pressure wave from the pump end of the injection line after nozzle needle closing.

As we introduced an extended SAC nozzle model with cavitation, it might be interesting to watch the cavitation start and duration. For this, the cavitation switch and cavitation factor are plotted in *Figure* 6-35, bottom graph. Cavitation switch stays zero as long as the needle does not lift up (i.e. there is no flow through the nozzle). When the needle moves (flow through nozzle), it attains value 1 for pure hydraulic flow (no cavitation) and value 2 for cavitating flow. The user may observe from *Figure 6-35* that cavitation starts very shortly after the needle lifts up and ends a few degrees of cam angle before the needle closes. The cavitation factor is given by the ratio

$$C_{cav} = \frac{p_{sac} - p_{cyl}}{p_{sac} - p_{vap}}, \qquad (6.2.3)$$

where p_{sac} is the pressure in the nozzle sac, p_{cyl} is the cylinder pressure and p_{vap} is the fuel vapor pressure in the nozzle sac. Diesel vapor pressure is very low compared to fluid phase pressure and can usually be neglected. Hence the cavitation factor C_{cav} may vary between 0 and 1.

The detailed analysis of the calculation results is beyond the scope of this manual and is left as an exercise for the user. Remember that output parameters selected in **Element | Store Results** can be viewed and processed further with the GUI postprocessor Impress Chart.

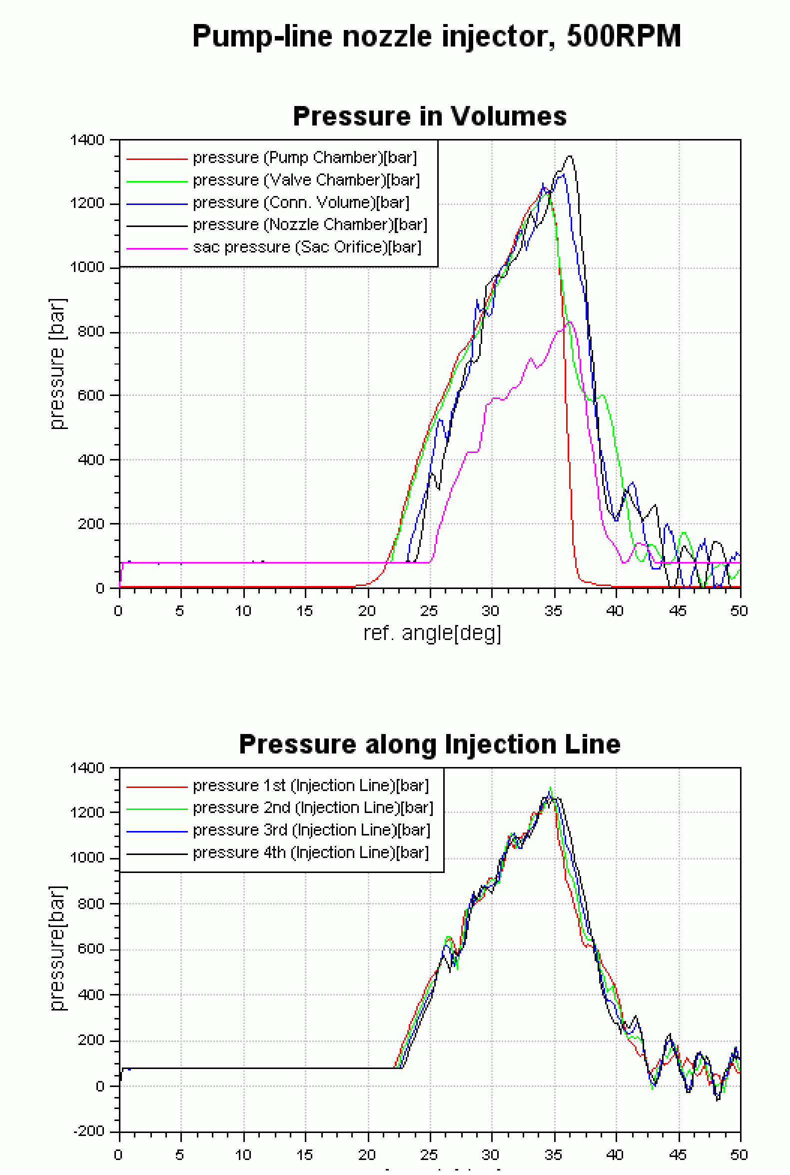


Figure 6-34: Pressures in Volumes and Four Cross-sections of Injection Line

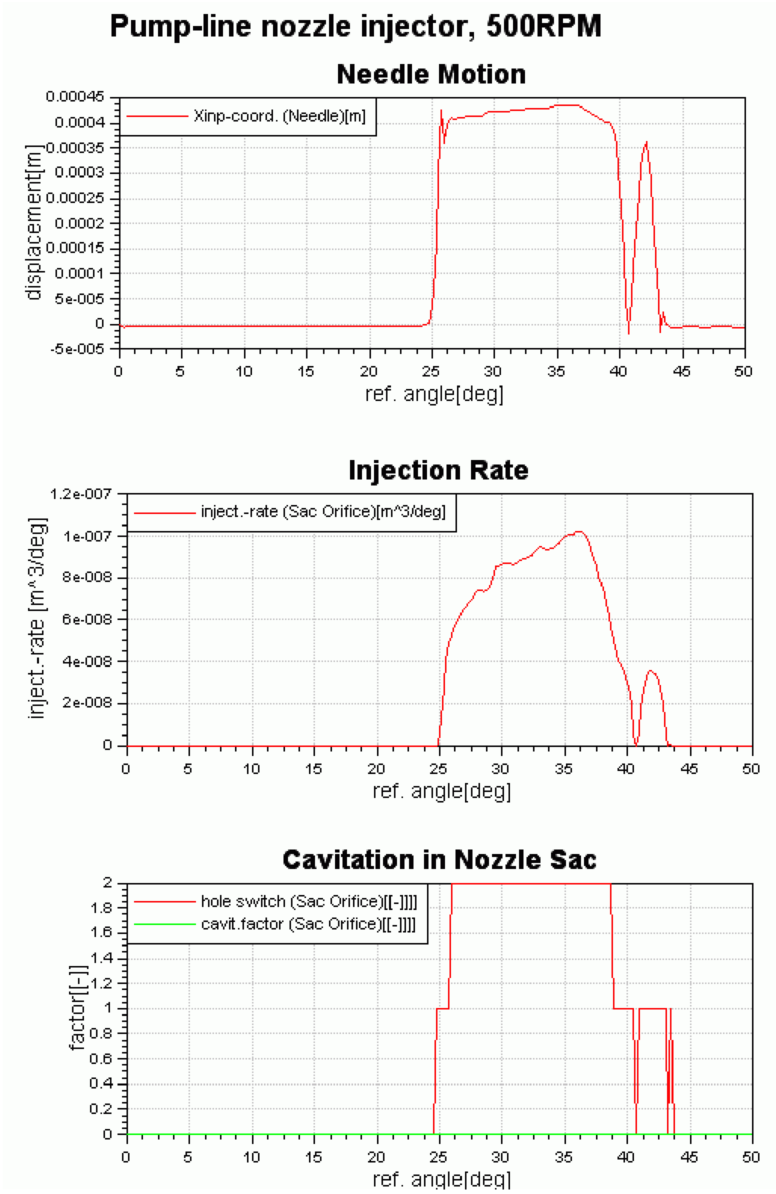


Figure 6-35: Needle Lift, Injection Rate and Cavitation Factors

6.2.4. Result Comparison

Here, we will show the result comparison between the two models for the flow calculation through nozzle spray holes: `inline_pump.hyd` (User-defined flow discharge coefficient) and `inline_pump1.hyd` (Program-calculated flow discharge coefficient). For input description of these models refer to *Section 6.2.1.6.* Although the theoretical approach behind each of them is quite different, both models give almost identical results (refer to Figure 6-36 and Figure 6-37). Of course, this does not imply that internal BOOST Hydsim model with **Program-calculated flow discharge coefficient** always yields exact results and is preferable. If the reliable values of the flow discharge coefficients can be attained (from measurements etc.), **the User-defined flow discharge coefficient** option should be used.

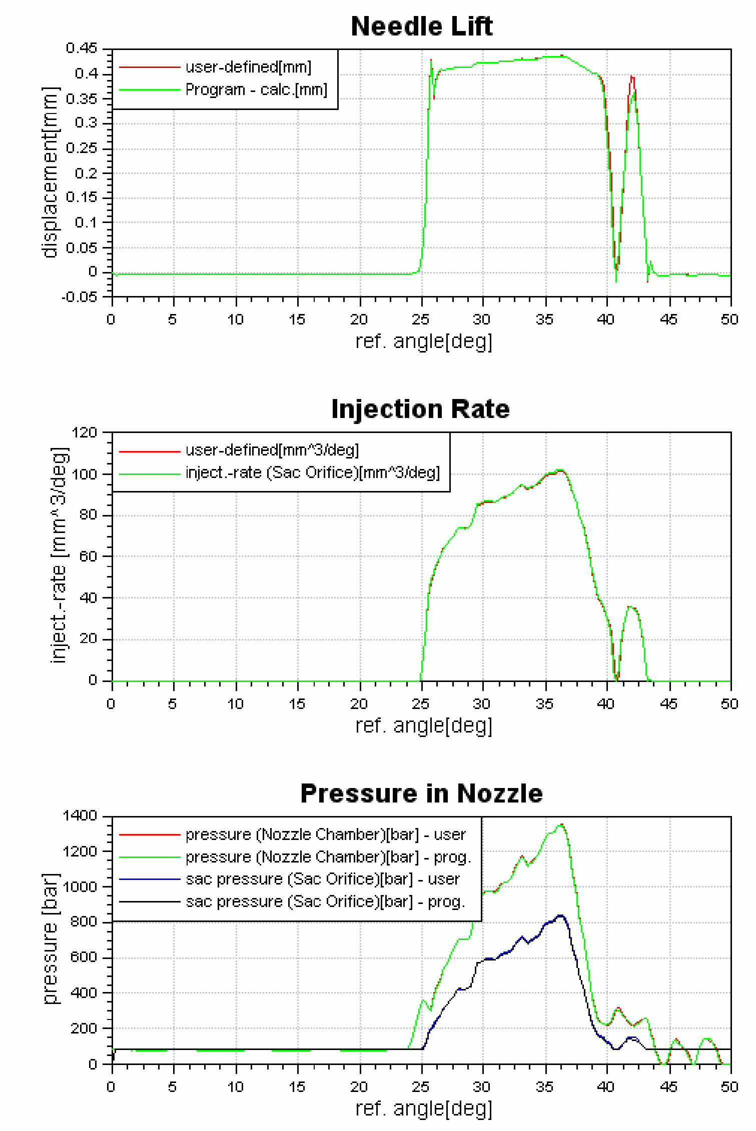


Figure 6-36: Needle Lift, Injection Rate and Nozzle Pressure Comparison

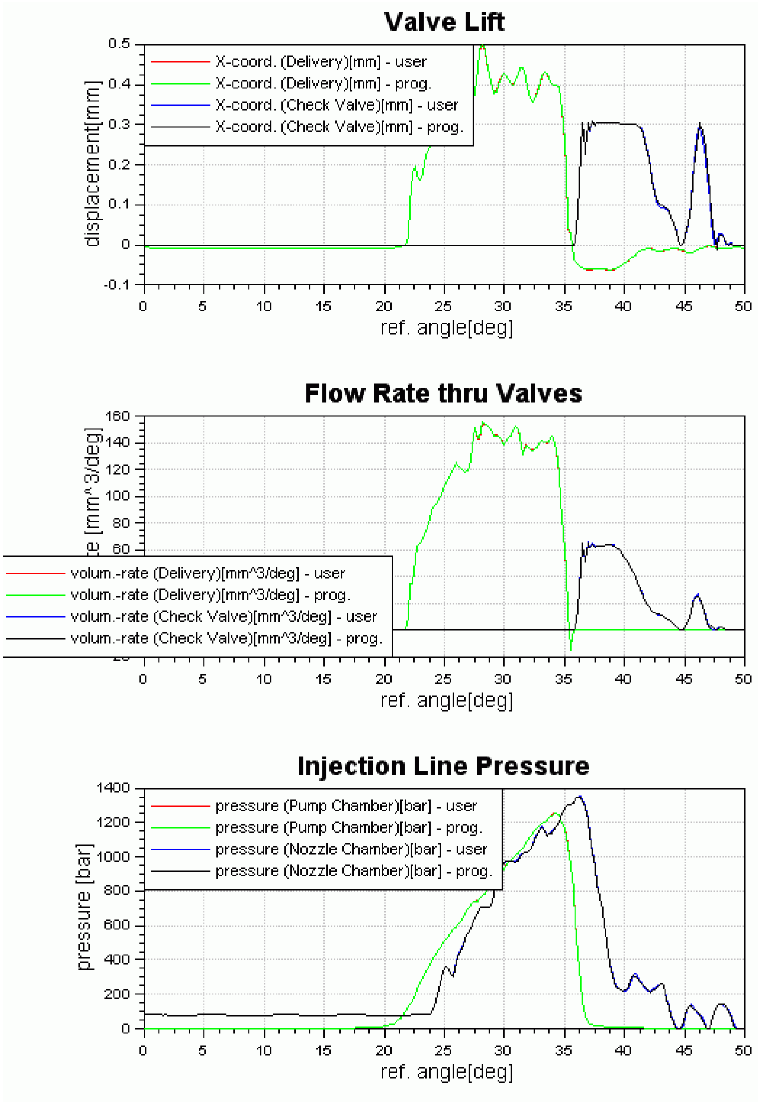


Figure 6-37: Result Comparison for Lift and Flow Rate through Delivery and Return-Flow (Check) Valve and Injection Line Pressure

6.2.5. Parameter Optimization (Search Adjust)

In this section it will demonstrated how to perform the parameter optimization with BOOST Hydsim program. **Optimization (Search Adjust)** is a powerful tool for reaching the required output target (e.g. injection timing or fueling) by adjusting the appropriate input parameters (e.g. effective plunger stroke).

Open file `inline_pump.hyd` and from menu **Model | Search Adjust** open **Search/Adjust Parameter – Set List** (*Figure 6-39*). In this dialog box click **Select** and the following dialog box will pop up:

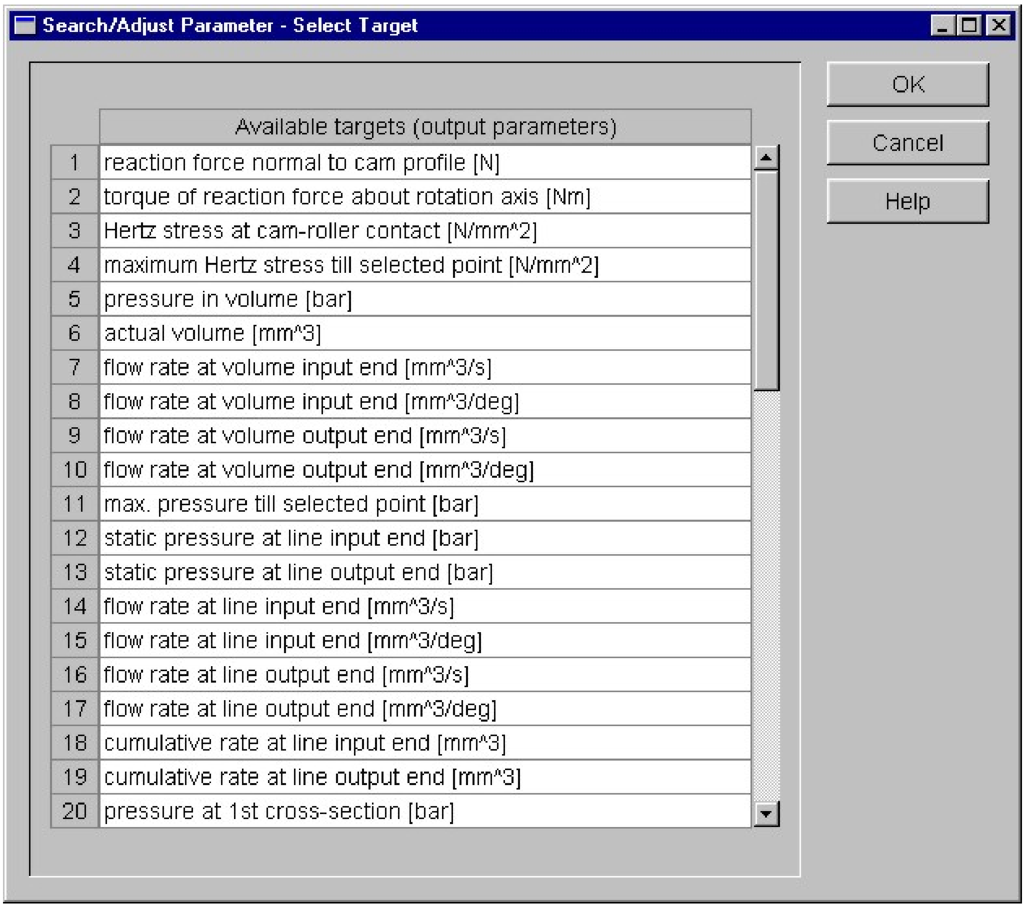


Figure 6-38: Select Target Dialog Box

From this list select "cumulative volume flow through nozzle orifice" (No. 51) and "max. pressure till selected point" (No. 11). Confirm selection by **OK**. Both selected items will be included into the Set List Dialog box as shown in *Figure 6-39*.

Note: "**Available Targets**" list in the Select Target dialog box contains all possible target functions for all elements in the current BOOST Hydsim model. Only one target can be selected at one time (for one optimization set).

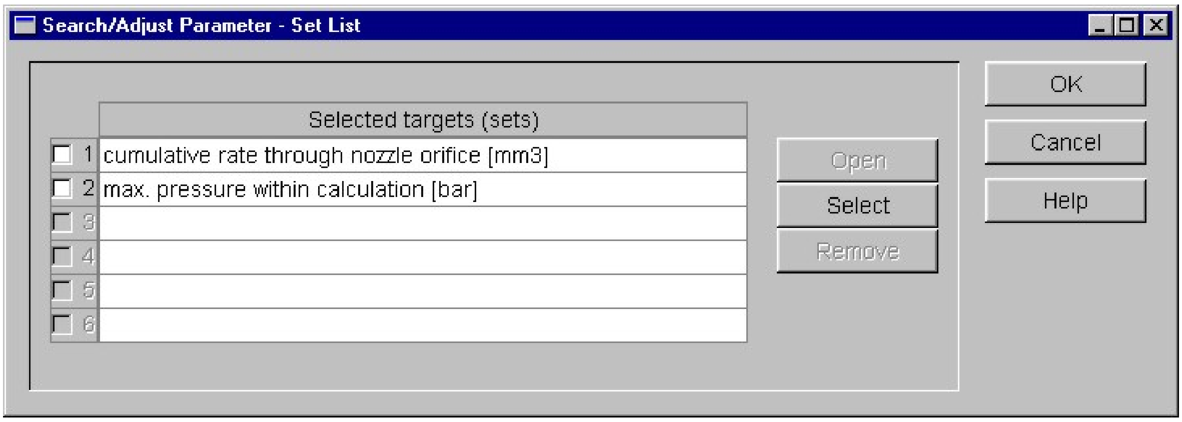


Figure 6-39: Set List Dialog Box

Click the check box in front of the field “cumulative volume flow through nozzle orifice”, then click on this field description line to highlight it blue and press **Open**. A new dialog box showing a definition of the Selected Set will be opened (*Figure 6-40*). Within this dialog, the user can define **Elements for target**, **Iteration control** and **Parameters for Adjustment**.

In the **List of elements for target** click the check box in front of the desired element to include it into the target function. In the **List of elements for adjust**, click the check box in front of the element “3 - Spill Port” and in its field, to highlight it blue. Then, in the **List of input parameters of selected element**, click the check box in front of “effective stroke of plunger [m]”. Set the initial step for this checked item to 0.001 m (this is the initial step for automatic iteration of the effective stroke of the pumping plunger, representing 1 mm of plunger lift, about its initial input value (previously input in the spill port element). Initial step is the amount by which the program will automatically change the initially input effective stroke for the first iteration, to produce a second result. It will then compare the difference of these to calculate its next effective stroke and will continue to iterate, within the number of defined iteration steps in the **Max. number of iterations** box and to within the accuracy for the fueling quantity, specified in the **Termination error** box.

Under **Iteration control**, values are input to define the required iteration control:

`Target value` is the desired value for the target (objective) function. In our case, that is the injection quantity or “cumulative volume flow through nozzle orifice” – set here to 1650 mm^3/injection.

`Max. number of iterations` - set here to 10. BOOST Hydsim will iterate up to 10 times to reach the required target (injection quantity).

`Termination error` is the percentage error defined by the user for the iterated output value, referred to the required target value – set here to 1%.

`Name for output files` is to be defined by the user. An original name, different from the main model filename (e.g. with an extension name describing the changed feature being modeled) must be used because the final iteration output results will be stored onto this file. Otherwise, confusion can occur: the output results of the initial run may be overwritten by the output results of the last iteration.

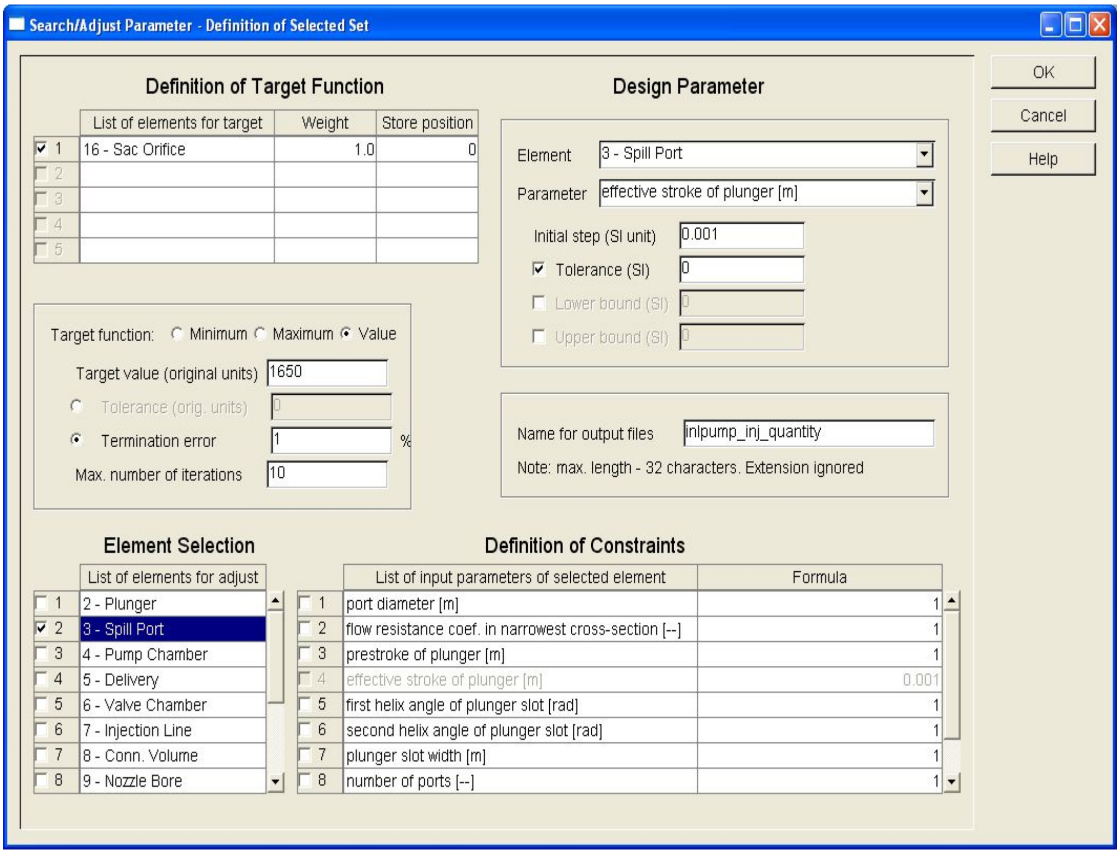


Figure 6-40: Definition of First Optimization Set

☞

Note: List of elements for target contains all elements of the current model, which have the chosen target function as output parameter (Store Results). To include the desired element into the target function, the corresponding check box in front of the element name must be clicked.

List of elements for adjust contains all elements of the current model, except those which have no optimized parameters (e.g. boundary conditions). To include the desired element into the optimization set, the corresponding check box in front of the element name must be clicked.

List of input parameters of selected element contains the input parameters of the selected element, which can be optimized. To choose the specific parameter for optimization, its check box has to be clicked and the initial step specified.

As we will see later in *Section 6.2.6*, optimizing on the fueling quantity alone will increase the injection duration and also the pump pressure. This will exceed the maximum possible pressure, therefore this system design is not acceptable. To alter it, the next Search/Adjust set has to be defined in the same manner with the pump pressure limit as the optimization target (refer to *Figure 6-41*).

Select "`4 - Pump Chamber`" as the target element, "`7 - Injection Line`" as the element for adjustment, "hydraulic diameter [m]" as its optimized parameter and set the Initial step to 0.0005 m (0.5 mm).

In **Iteration Control** set the Target value to 900 bar, Max. number of iterations to 8, and Termination error to 2%.

Replace the Name for output files to "`inlpump_ad2`".

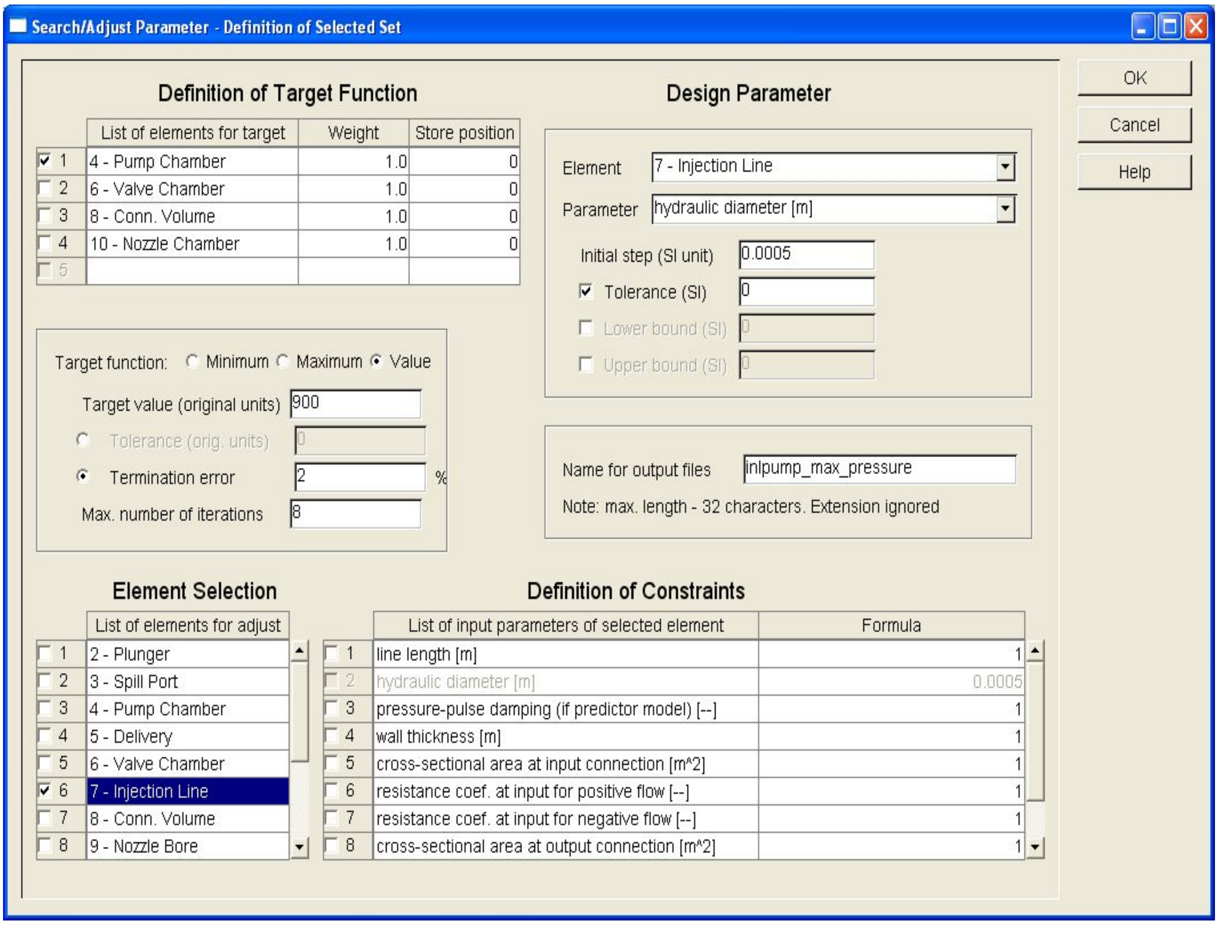


Figure 6-41: Definition of Second Optimization Set

Before confirming with **OK,** click the check boxes in front of both selected Sets (*Figure 6-39*).

Save this file under a different name, for instance `inline_pump_ap.hyd` and start the calculation. **View Logfile** should show the following:

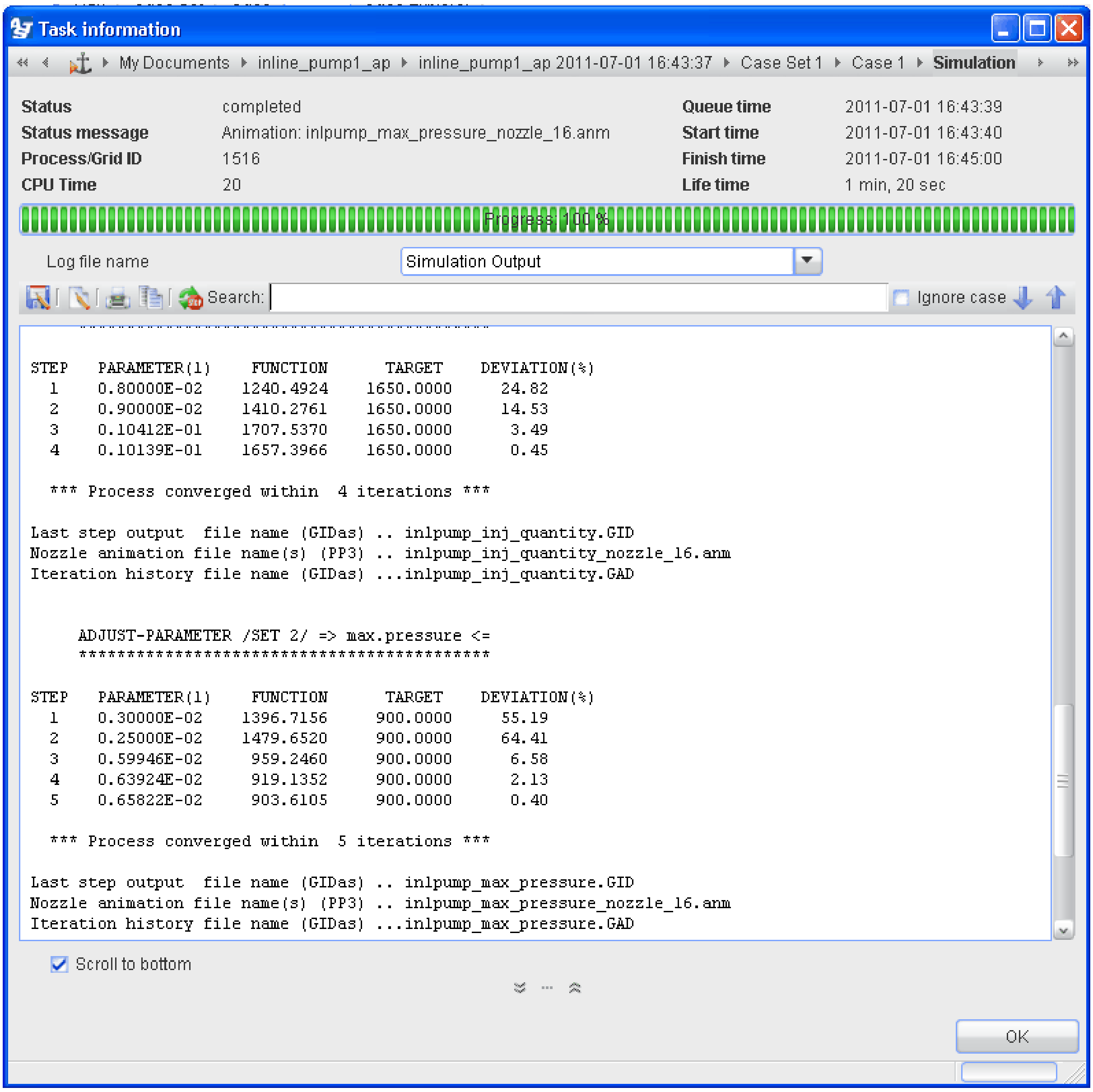


Figure 6-42: View Logfile with Search/Adjust/Parameter Results

BOOST Hydsim generates three GIDas files with extension *.GID containing the calculation results (for the initial run and last iteration of each optimization set) and also two small GIDas files with extension *.GAD files containing the iteration history of each optimization set in the case-specific sub-directory:

<project_dir>/BOOST Hydsim/<model_name>.<case_name>

Note: If the optimization procedure converges within the specified max. number of iterations, the message *** Process converged within n iterations *** will pop up in the View Logfile window where n is actual number of iterations. Otherwise the message *** Process did not converge within n iterations *** will appear, which implies that the target value was not reached within the specified accuracy (termination error). Nevertheless, the last iteration results are stored on the GIDas file "Name for output files.GID". Iteration history is stored on the GIDas file "Name for output files.GAD".

6.2.6. Optimization Results

To view the optimization results, open Postprocessor Impress Chart, refer to *Section 4.5*.

Load results from specified directory. The user may observe that it is possible to expand the **Search Adjust** folder (*Figure 6-43*).

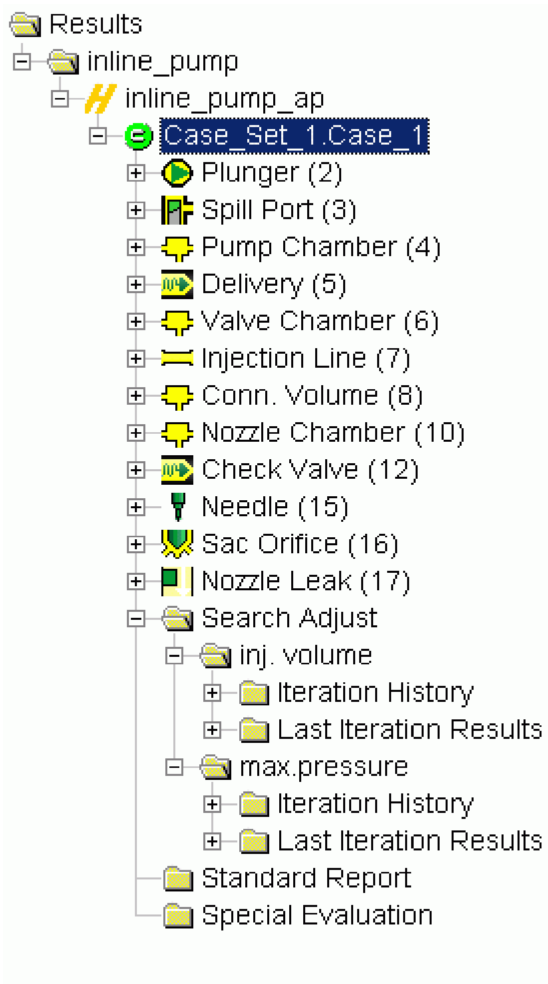


Figure 6-43: Expanded Results Tree with Search/Adjust Folder

Initial calculation results are located in the upper part of the result tree (**inline_pump**). The optimization results (**Last Iteration Results** folder) and the iteration history (**Iteration History** folder) are available under the sub-folders (**inj. volume** - 1st optimization set and **max. pressure** - 2nd optimization set). This structure allows the direct comparison of all results (with and without optimization).

The user can use the **Templates** layers to view Pressure in Pump and Nozzle Chambers, as well as Needle lift and Injection Rate. Open **Templates** Tab folder and double click on the **BOOST Hydsim Templates** folder. From the expanded folder tree select e.g. inline_pump1. The following layers will be created, as is shown in *Figure 6-44*.

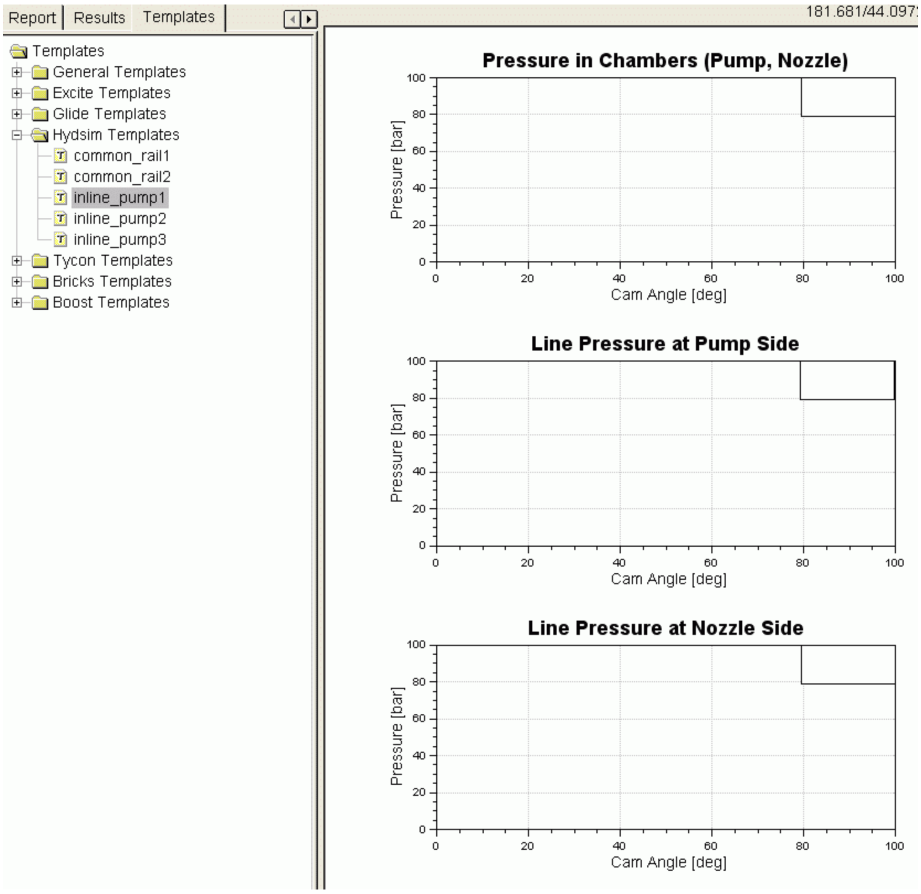


Figure 6-44: Template Layers

The user can easily modify the layers and create its own template. Assume we want to plot the pressures in pump and nozzle chamber for all calculation cases (initial model and both optimization runs) on one template and the needle lift and injection rate on the other. With a minor editing effort, we create first the template with two layers as shown in *Figure 6-45*. For the needle lift and injection rate, we create another Report Page (refer to *Section 4.5.1*) and from the expanded folder tree in Templates Tab folder select **inline_pump2**. All we need to do now is to place the appropriate curves onto the proper layers. The final layout is shown in *Figure 6-46*.

All created templates can be saved under **User-defined Templates**.

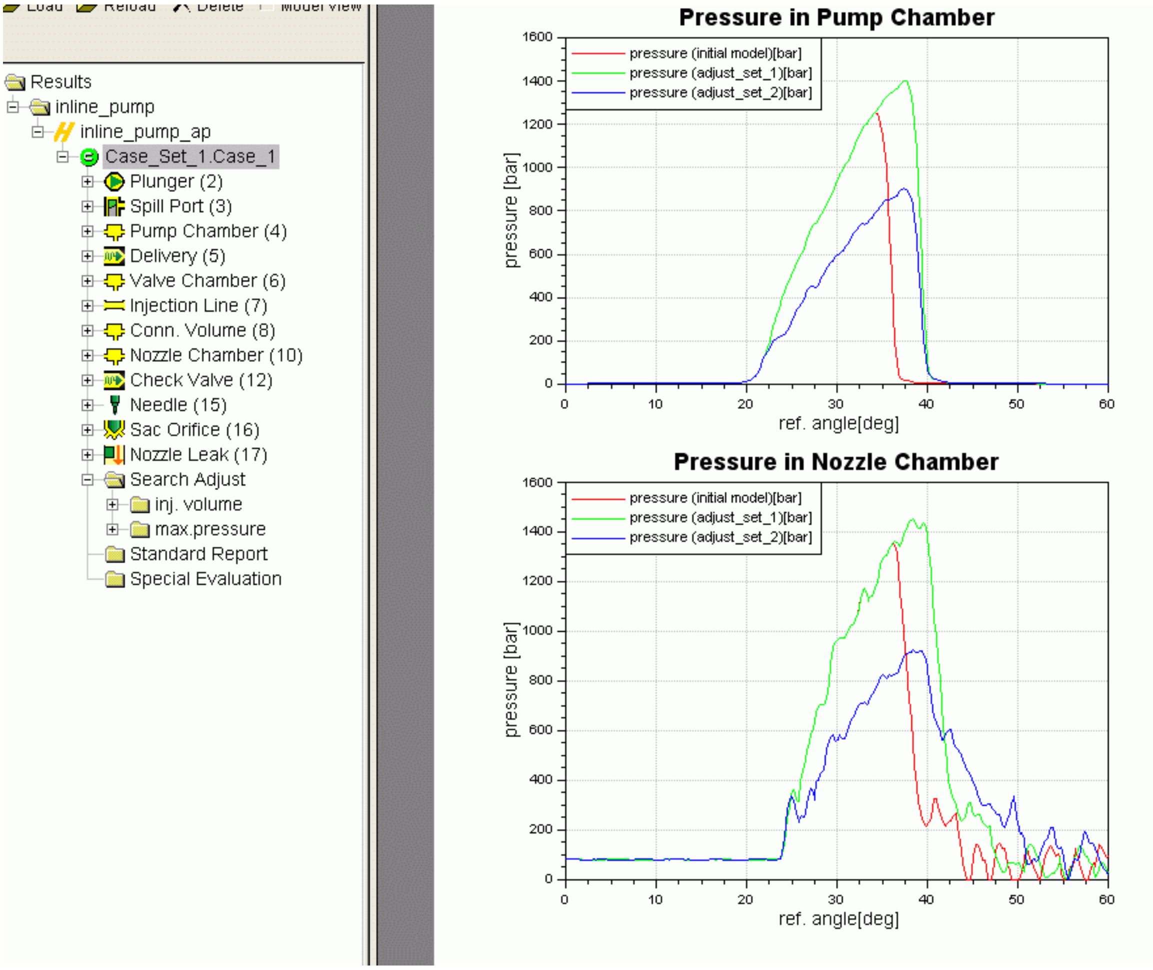


Figure 6-45: Pressure in Pump and Nozzle Chambers (User-defined Template)

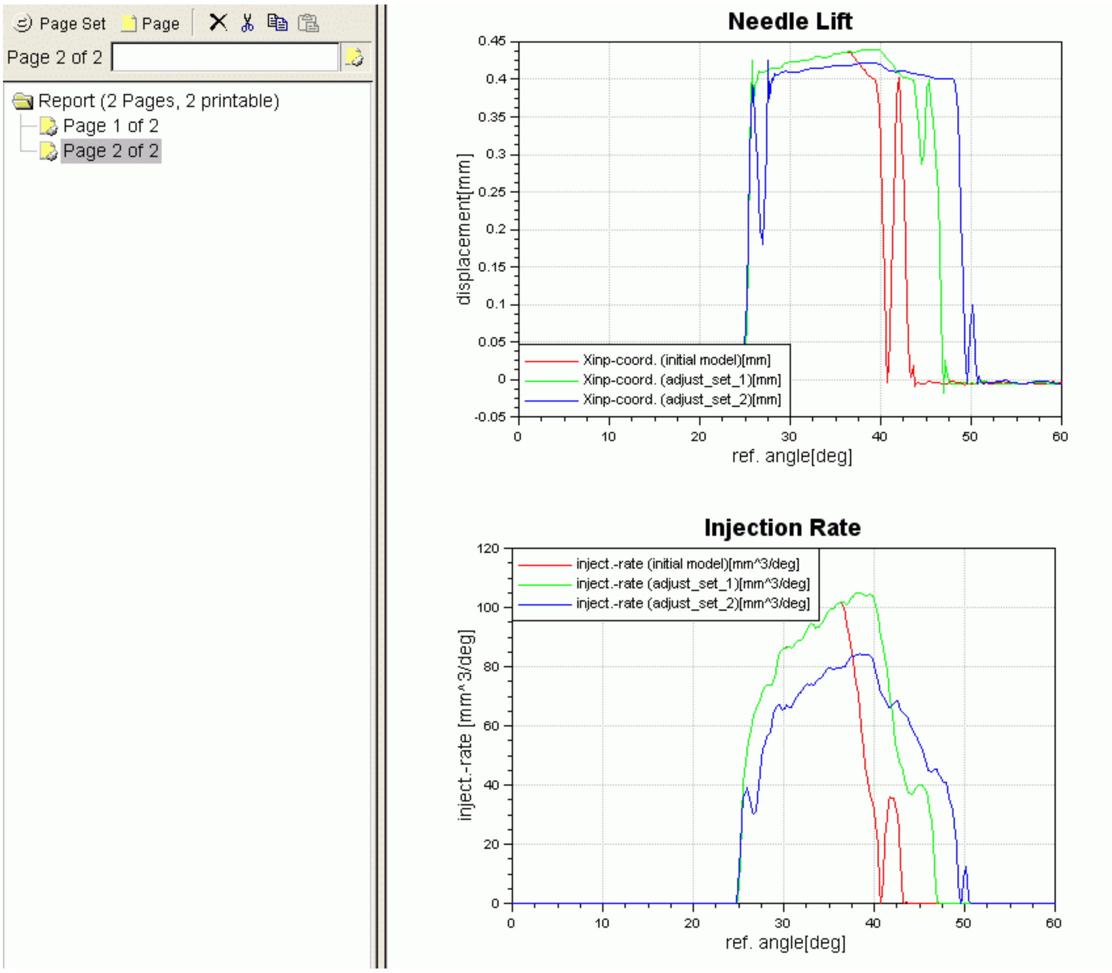


Figure 6-46: Needle Lift and Injection Rate

Figure 6-45 shows that in the process of the 1st optimization (adjust parameter) procedure the pressure in pump and nozzle chamber considerably increased because the effective stroke of the plunger increased from 8 mm to 10.05 mm. In this way, the required fueling of 1650 mm^3/stroke was achieved (against the initial value 1248 mm^3/stroke). However, the pump pressure limit was highly exceeded (1370 bar against allowed 900 bar), therefore this system design could not be accepted. To reduce the pump pressure, an attempt to adjust the injection line diameter was made in the 2nd optimization set. The effective plunger stroke was kept unchanged. Optimization results showed that the pump pressure could be reduced to 900 bar by increasing line diameter from 3 mm to 6.3 mm. Obviously the needle lift and injection duration increased further as shown in *Figure 6-46*. However, at the same time the injection quantity dropped to 1535 mm^3/stroke. This shows the difficulty of the one-dimensional optimization procedure where the change of one optimized parameter may alter the target value attained in the previous optimization runs.

6.2.7. Animation

Open again file inline_pump.hyd.

To perform Animation of the Nozzle Flow, refer to *Sections 5.1.8*.

The animation file inline_pump_nozzle_17.mod is stored in the case-specific subdirectory: <project_dir>/BOOST Hydsim/<model_name>.<case_name>

Open the Input Data dialog box of SAC Nozzle Orifice (refer to *Figure 6-32*) and select the appropriate type for **Spray Calculation**, for instance:

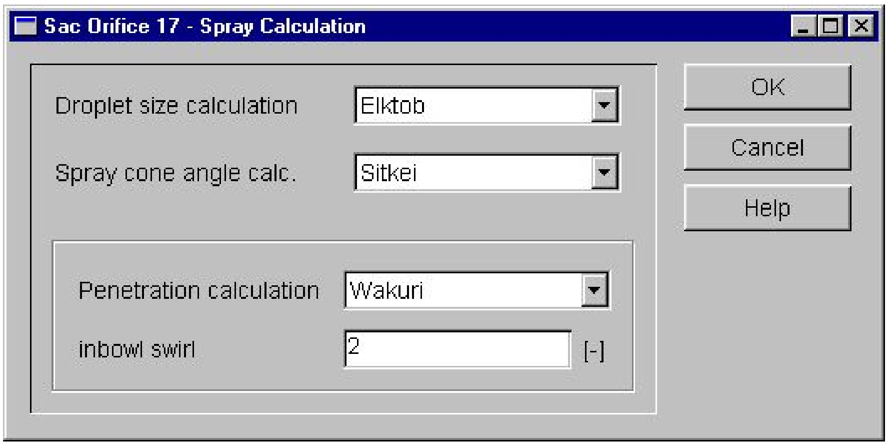


Figure 6-47: Spray Calculation Dialog Box

6.2.8. Running Animation

To view the animation results, refer to *Section* 5.1.8.

Open the animation file inline_pump_nozzle_17.mod and perform animation.

Note: Animation can be performed for the initial model and for each optimization (search adjust) set. Animation files of optimized models are inlpump_ad1_nozzle_17.mod and inlpump_ad2_nozzle_17.mod (refer to specified names for output files in the Search/Adjust dialog box - *Figure 6-40* and *Figure 6-41*).

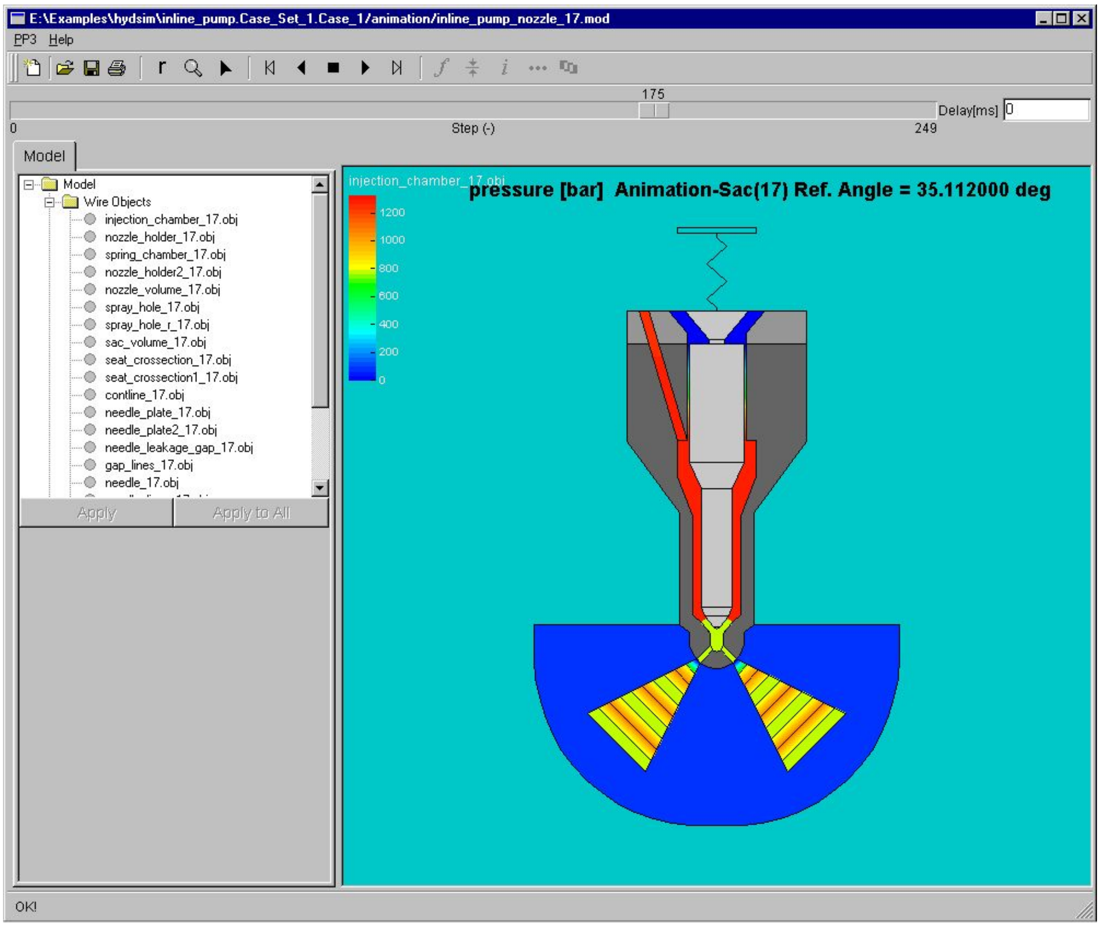


Figure 6-48: Nozzle Animation Screenshot (File `inline_pump_nozzle_17.mod`)

Figure 6-48 shows the animation screenshot for the first optimization set.

The user may observe the difference in relation to the animation of the **common rail** example (*Figure 5-25*). For **conventional injection systems**, only the nozzle animation is shown: it covers the pressure distribution, needle motion, nozzle leakage and spray (if defined). For **common rail systems**, the pressure in the control chamber and the motion of control piston is plotted additionally. Note that animation does not aim to represent the real design of the nozzle, but serves only for the fast visualization of the calculation results.

Refer to the PP3 section of the GUI Users Guide for more information.

6.3. Electronic Unit Pump - Line - Injector

In this example, the modeling of a conventional pump-line-nozzle system, using an electronically-controlled pump and conventional injector, will be demonstrated. The system size pertains to a typical medium-speed diesel engine as could be used on a power generation, ship propulsion or locomotive application.

The system is outlined in *Figure 6-49*, works at around 1200 bar peak pressure and consists of:

- a falling-rate fuel cam having a lift flank duration of 50 cam degrees
- a fuel pump having a 22 mm diameter plunger and a fast solenoid-controlled on/off valve which controls both the fuel timing (start of injection) and the fuel delivery (duration of injection). This valve is 'normally open' and is energized closed to create injection. Plunger displacement flow spills directly back to the pump gallery when the pump is not pressurizing fuel for injection. No delivery valve is fitted to this pump. The plunger has no fuel quantity control helix and can be considered as a simple piston, with leakage
- a high-pressure line and a 'quill pipe' connecting the pump to the injector
- an injector having a conventional shim-set nozzle spring, with a transfer block between the nozzle body and the nozzle holder body, with nozzle needle/body leakage to a back-leak passage in the engine cylinder head

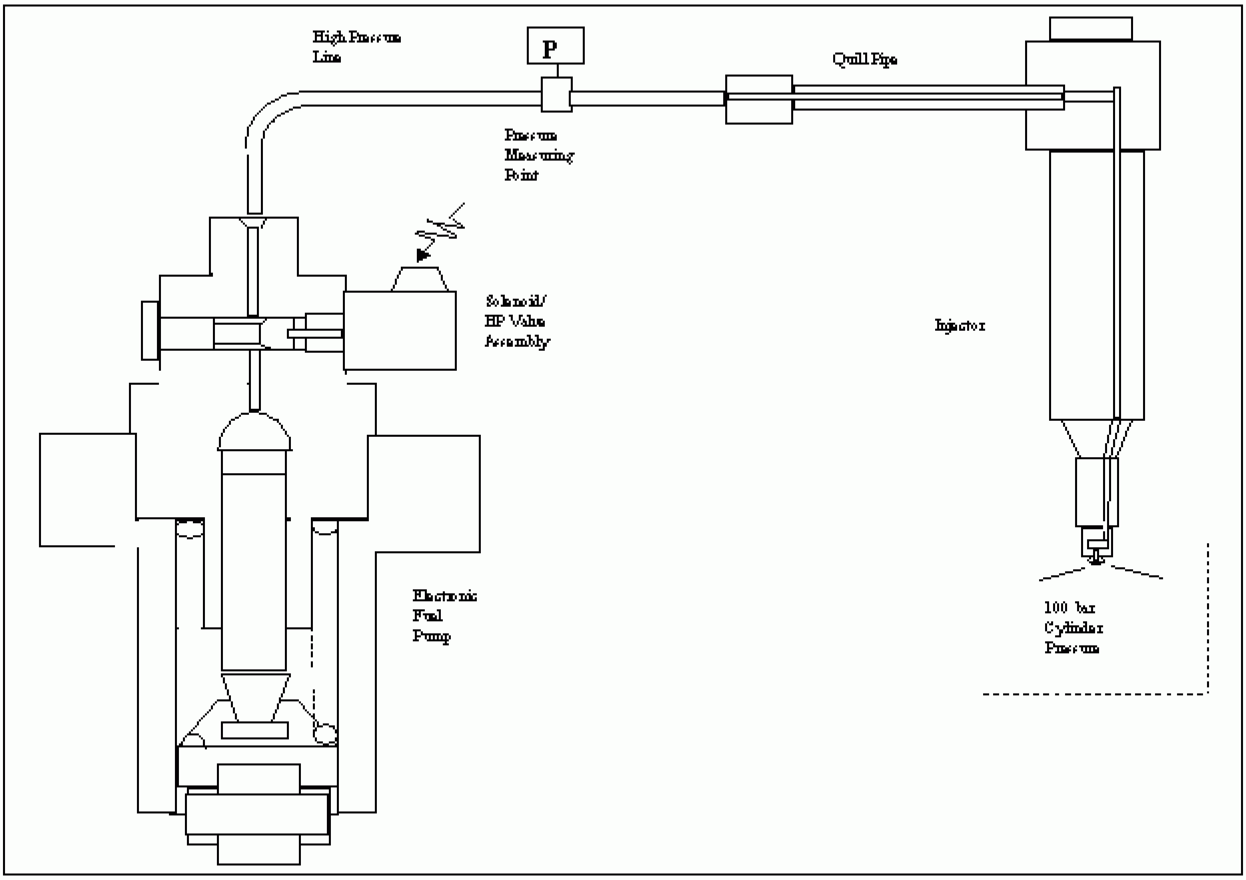


Figure 6-49: Schematic of an Electronic Pump-Line-Injector System

As before, the fuel pump is driven by Cam Profile 1which is a mechanical boundary defining the plunger kinematic excitation (acceleration). Similar cam/roller follower (pumping plunger) stiffness and damping features apply and are modeled using a mechanical connection.

The pumping Plunger 2 connects to a Pump Volume 3 which splits to a forward delivery drilling in the pump head HP Bore1 5 and a plunger/ barrel leakage path Plunger Leakage 4, connecting to Leak Pressure 20 set to the 8 bar fuel gallery feed pressure. Note again the special connection (green) between the Plunger and the Leakage element. The pump delivers its fuel into MesVol1 6 which is a volume element added into the model to represent a pressure measurement point at the pump outlet connection. This volume feeds the first section of a split high-pressure line InjLine1 8, which is again divided at a pressure measuring point MesVol2 23.

☞ **Note:** The latest version of BOOST Hydsim does not generally require Volume elements between the line-line, line-orifice or similar elements. These can now be connected to each other directly. However, if specific features such as pressure measuring points on the engine are to be modeled, it is recommended that specific Volume elements are placed in these positions in the model. One of the more complex Line elements having the capability to be split into adequate divisions, so that measuring points could be designated at specific line segment junctions, could be used but this can lead to calculation 'noise'. Also it is easier for the user and others to see the specific modeled measurement points in the model layout.

Such 'measuring' volumes are given a volume input value equal to the cross-sectional area of the line multiplied by the diameter of the line bore and rounded-up to a sensible value. This ensures a minimum volume but with no restriction to flow through it, compared with the line each side of it. Such volumes can be accounted for by accordingly reducing the line length adjoining them, if necessary, although their volume is usually insignificant in the total system volume and will have an insignificant effect on the results.

The second pressure measuring point connects to the other section of the high-pressure line InjLine2 9, which in turn connects to the injector via the injector body drilling NH Bore 17.

The rest of the injector layout is much as described before, including a leakage path from the nozzle needle and a cylinder pressure input (here a constant 100 bar) onto the needle tip. It is not further described here except to note again that a combined mechanical/pressure boundary is used for the needle leakage and needle lift stop features, also a special (green) connection is again required between the needle and leakage to define their positional dependence.

The control of the fuel pump is modeled using a time-controlled throttle, Solenoid Valve 11. This is described in more detail later.

The Solenoid Valve has a parallel Solenoid Leakage 7 element across it, connecting to a small Spill Volume1 12. This in turn connects to an Orifice1 14, representing a restriction to the spill into the pump gallery, whose large volume is represented by Spill Volume2 13.

A further Orifice2 15 represents a restriction from this pump fuel feed/return gallery to the main engine feed rail, installed to retain pressure within the pump to prevent cavitating conditions and to limit spill pressure peaks in the engine low-pressure feed rail. This orifice connects to the 8 bar feed rail of the engine. Note that the pumping element is filled via flow through the solenoid-operated high-pressure valve whilst it is in the open position. The plunger is falling on the return flank of the fuel cam. Hence flow occurs in both directions through this spill/fill path but the hydraulic connections here are shown only in the spill direction - this makes no difference to the ability to fill the pump in the modeling of it.

6.3.1. Creating the Model

The complete BOOST Hydsim model of the electronic pump-line-injector system depicted in *Figure 6-50* consists of the following elements:

- Cam Profile **(CAM/Cam Profile)**
- Plunger **(PUMP/Plunger)**
- Pump Volume **(VOLUME/Standard)**
- HP Bore1 **(LINE/Laplace Model)**
- MesVol1 **(VOLUME/Standard)**
- InjLine1 **(LINE/Laplace model)**
- MesVol2 **(VOLUME/Standard)**
- InjLine2 **(LINE/Laplace model)**
- NH Bore **(LINE/Laplace model)**
- Nozzle Volume **(VOLUME/Standard)**
- Spray Holes **(NOZZLE/Sac -basic model)**
- Cylinder Pressure **(BOUNDARY /Pressure)**
- Needle **(NEEDLE/Standard-up-to-date model)**
- Nozzle Holder **(BOUNDARY/Hydromechanical)**
- Needle Leakage **(LEAKAGE/Annular gap)**
- Plunger Leakage **(LEAKAGE/Annular gap)**
- Leak Pressure **(BOUNDARY/hydraulic)**
- Solenoid Valve **(THROTTLE/time-controlled)**
- Solenoid Leakage **(LEAKAGE/Annular gap)**
- Initial spill volume **(VOLUME/Standard)**
- Orifice1 **(ORIFICE/General)**
- Spill Volume2 **(VOLUME/Standard)**
- Orifice2 **(ORIFICE/General)**
- Feed Pressure **(BOUNDARY/hydraulic)**

The complete BOOST Hydsim model for this system looks as follows (*Figure 6-50*):

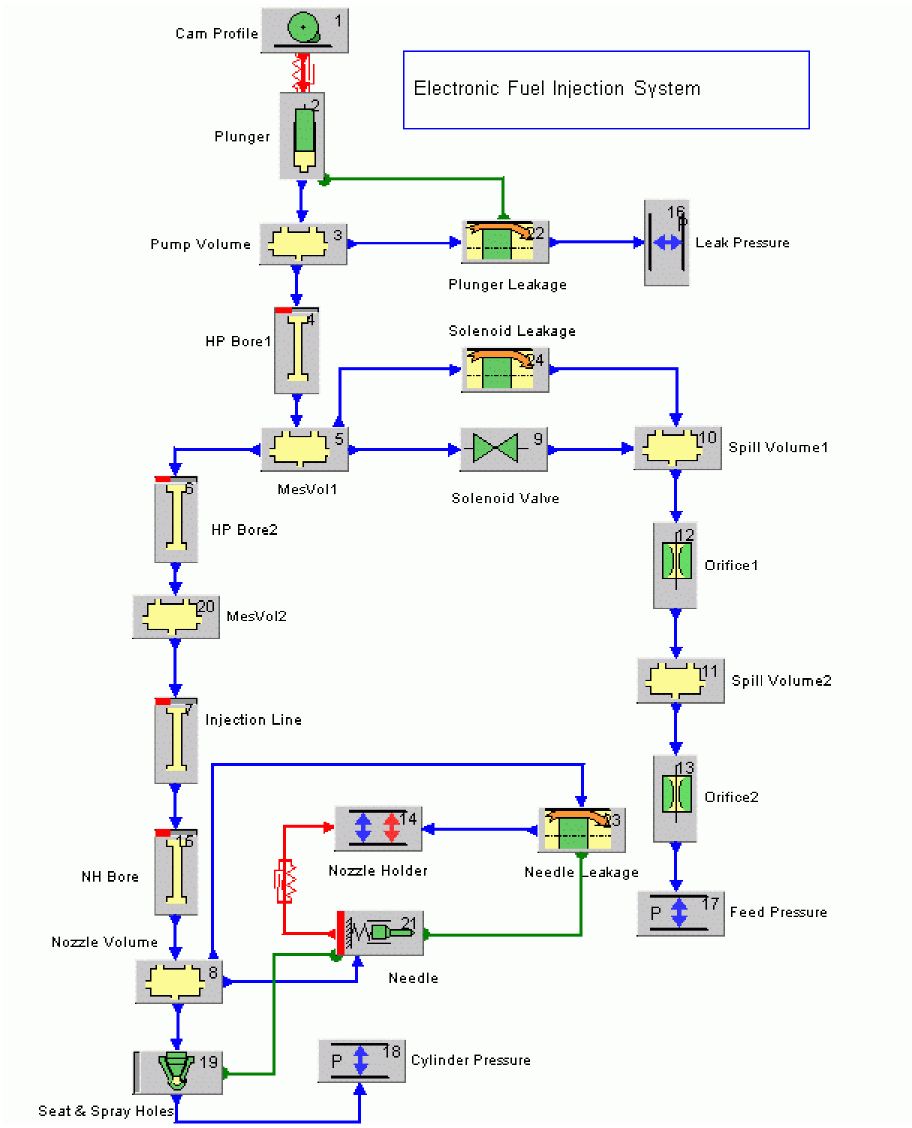


Figure 6-50: BOOST Hydsim Model of the Electronic Pump-Line-Injector System

Only the new features in this model will be discussed further here. These are primarily the use of the time-controlled throttle for the high-pressure solenoid valve control, for both fuel injection timing and fuel delivery quantity and its input parameters and optimization (search/adjust) features.

6.3.1.1. Solenoid / Control Valve

Figure 6-51 shows the input dialog box for the Solenoid Valve (time-controlled throttle) element.

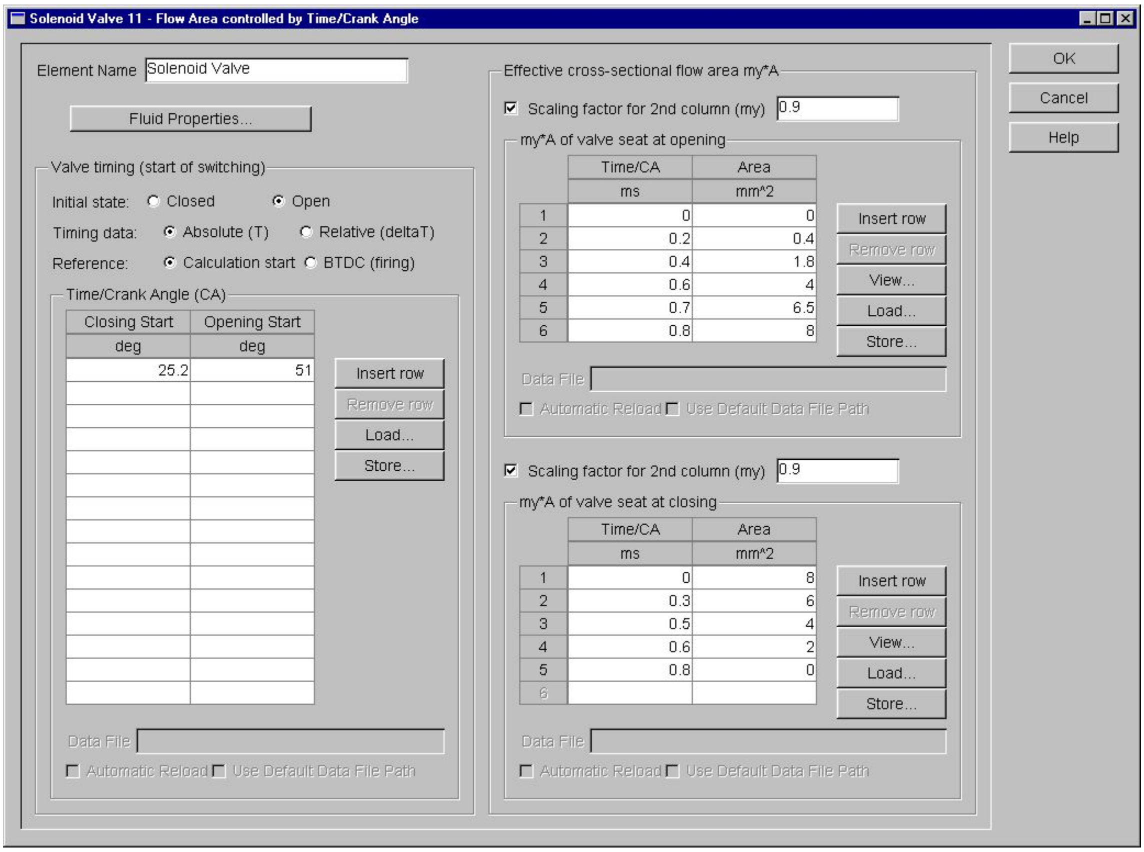


Figure 6-51: Input Dialog Box of the Solenoid Valve (Time-controlled Throttle)

This dialog box has the following input features:

- Valve timing (start of switching) - initial state, closed or open (here open)
- Angle (in degrees CRANK) for the required start of the solenoid valve closure (to start injection) - note that there are both electronic and hydraulic delays in the injection. This value will be obtained from a knowledge of the programmed timing of the fuel pump, which would be input into the fuel timing map of the engine's fuel system electronic control unit (ECU), here for the full-load/speed setting for this system
- Angle (in degrees CRANK) for the required start of solenoid valve re-opening for the end of injection (again electronic and hydraulic delays apply, so the valve will, in reality, be switched earlier than would be expected, to account for these delays, otherwise excessive fueling quantity would result. This again is a programmed value within the fuel quantity map within the ECU of the fuel system control. Here we shall iterate to obtain the correct solenoid closed duration time, to obtain the required full-load fuel delivery

- Two tables of solenoid actuation time for closing and then re-opening, to model the electronic delays. The hydraulic delays are modeled by BOOST Hydsim from all system input features. These values are either obtained from the fuel injection equipment or solenoid/valve assembly supplier or from knowledge of the measured actual operating characteristics of the solenoid/ high-pressure valve assemblies.

Note that the Scaling Factors (flow discharge coefficients) here are both set to 0.9 because the areas within the tables are actual geometric flow areas and take no account of flow loss through the valve seat.

Refer to *Section 6.2.5* for performing Parameter Optimization, here with the following parameters:

`Selected targets`: "cumulative rate through nozzle orifice"

`Element for target`: no. 21 – Spray holes

`Element for adjust`: no. 11 – Solenoid Valve

`Parameter for Adjustment`: switching time T2 (initial step 0.002 s)

`Target Value`: 2427 (mm^3 / injection delivery)

`Max. number of iteration`: 6

`Termination error`: 1%

6.3.2. Calculation Results

We shall show results for the main injection features of such a system at the rated engine condition. These would normally include:

- Nozzle needle lift
- Pressure in the pumping chamber
- Pressures at any points in the system which may be used later during test stand or engine testing of the fuel system, e.g. in the head of the fuel pump, along the injection line or in the head of the injector body
- Injection rate
- Any other features of interest, such as valve movements in the fuel pump

To view the calculation results, select **Show Results** from **Simulation** Pulldown menu, to open Impress Chart window with the right model directory, as shown below.

Figure 6-52: Show Results Menu

Figure 6-53 shows the nozzle needle lift and injection rate. A clean needle lift having a slight overshoot of the needle at initial full lift and a slight undershoot at closure is shown - these features confirm a good modeling of the needle element stiffness.

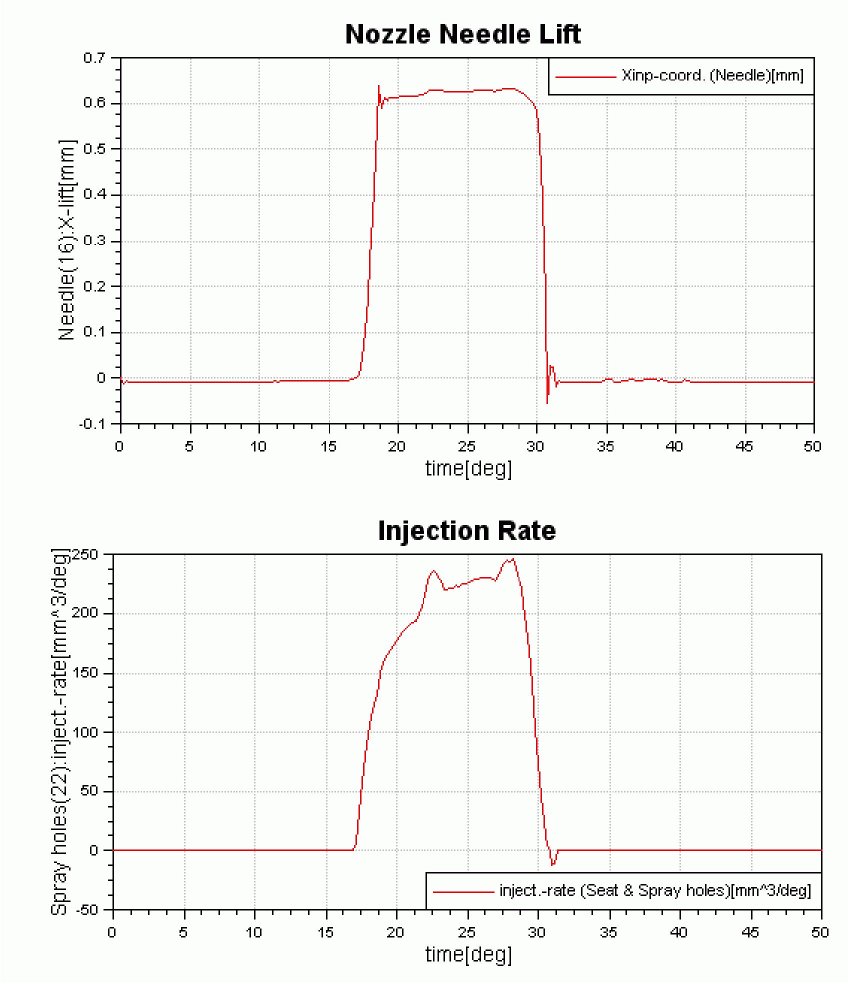


Figure 6-53: Nozzle Needle Lift and Injection Rate

Figure 6-54 shows pressures in the injection pump chamber, in the measuring point along the high-pressure line and in the injector nozzle gallery. All have very similar peak values, confirming minimum pressure loss or friction through the line chosen. The spill rate is determined by the solenoid valve flow area and after-injection wave features are determined by the spill path orifices and volumes. Because there is no delivery valve in this pump, these after-injection features cannot be altered in any significant way and no residual pressure in the line is produced (the line pressure drops down to the feed pressure value of 8 bar).

The plunger velocity and Hertz contact stress are also shown in this diagram because it is important to relate the pumping pressure with the plunger velocity and to check the maximum cam/roller contact stress. In this way, we can confirm that the pumping both starts at an adequate point on the velocity diagram and that the pump spill is well underway adequately before the 'nose' of the fuel cam, to prevent over-stressing the fuel cam-roller contact.

Note that the oscillations in the Hertz contact stress and Plunger velocity curves result mainly from the small damping value of the mechanical connection between Cam Profile and Plunger and are not physically meaningful. Inputting a significantly higher damping value would damp these out.

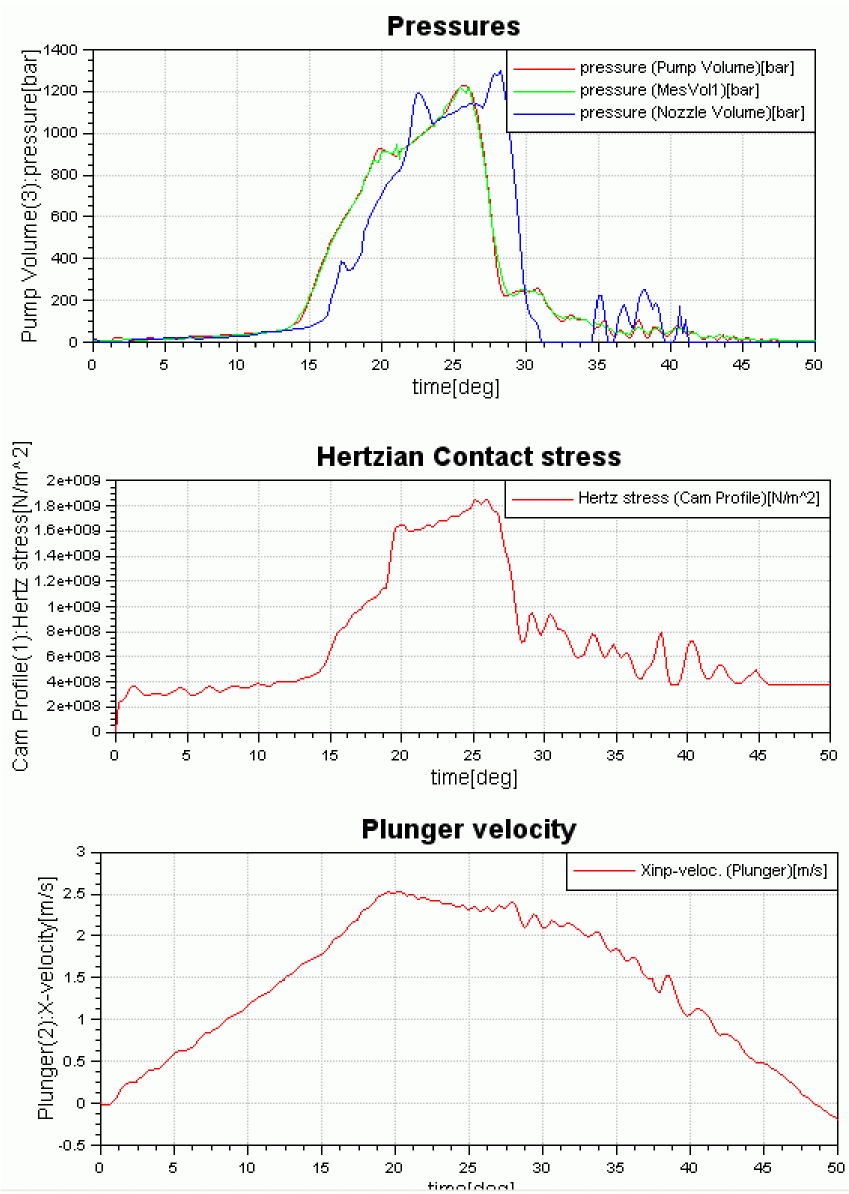


Figure 6-54: Pressures through the System, Hertz Stress and Plunger Velocity

Figure 6-55 shows the Solenoid Valve flow area, giving the 'theoretical' start and end of injection (no hydraulic delays so far accounted for), also the spill path features. Here, the initial flow out through the spill valve is shown, as the pumping plunger rises and displaces fuel to spill, before the solenoid valve is energized to be closed, also the effects of spill from high pressure at the end of injection.

Note the 80-90 bar peak pressure spill pulse created in the pump gallery at main spill.

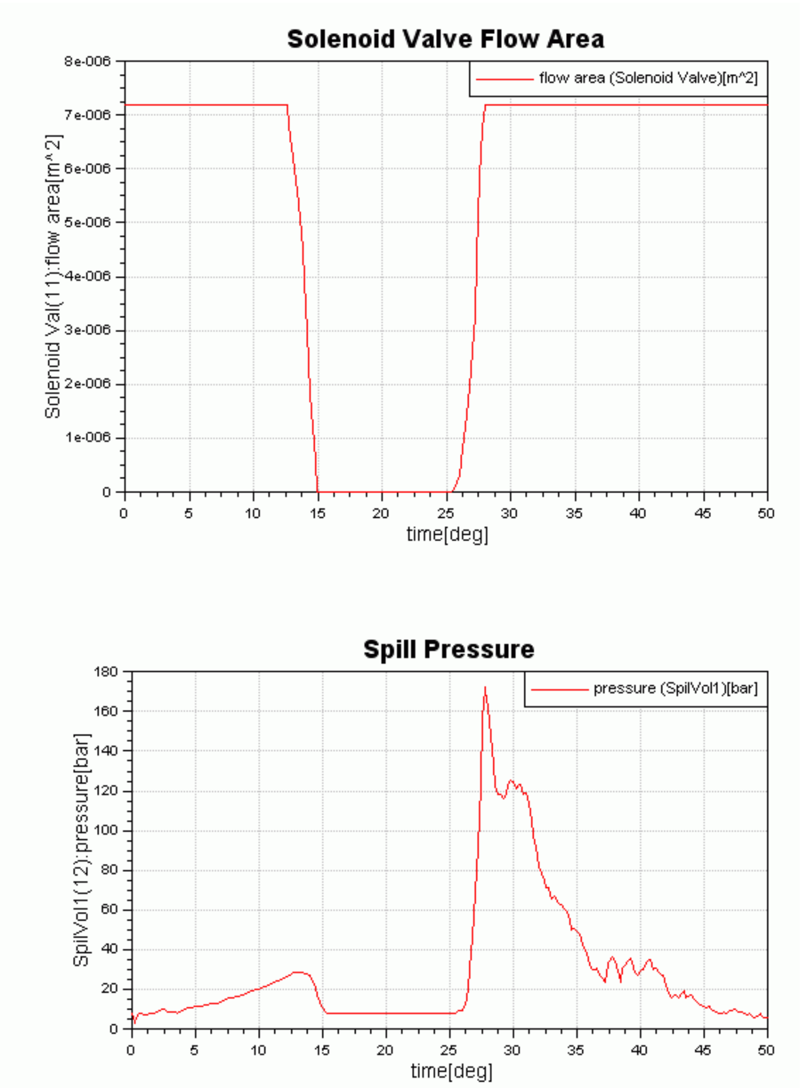


Figure 6-55: Flow Area through the Solenoid Valve and Spill Pressure

6.3.2.1. Engine load cases

Create three Cases for three engine load and speed cases. In **Calculation Control** set `Reference/Engine speed` to `500/1000 rpm` (first case - 100% load), `420/840 rpm` (2nd case - 55% load) `and 310/620 rpm` (3rd case - 25% load). Save the model under a different name, for instance `electronic_pump_load.hyd`.

Figure 6-56 shows the rated engine load and speed case, compared with the 55% and 25% load cases. As engine load is reduced, the engine (fuel camshaft) speed reduces, in a manner similar to a cubic power law. The figure shows that the injection occurs earlier as load and speed reduce. This is because no account of the injection timing requirement has been made here (the solenoid valve closure time T1 is left constant), however, the wave travel time through the injection system is constant (dependent only on the speed of sound in this particular fuel). Therefore as engine speed reduces, the pressure wave reaches the injector earlier at lower speed than it does at higher speed. In reality, this would be accounted for by mapping the injection start timing later in the fuel pump controller than is done here for the 25% and 55% load cases.

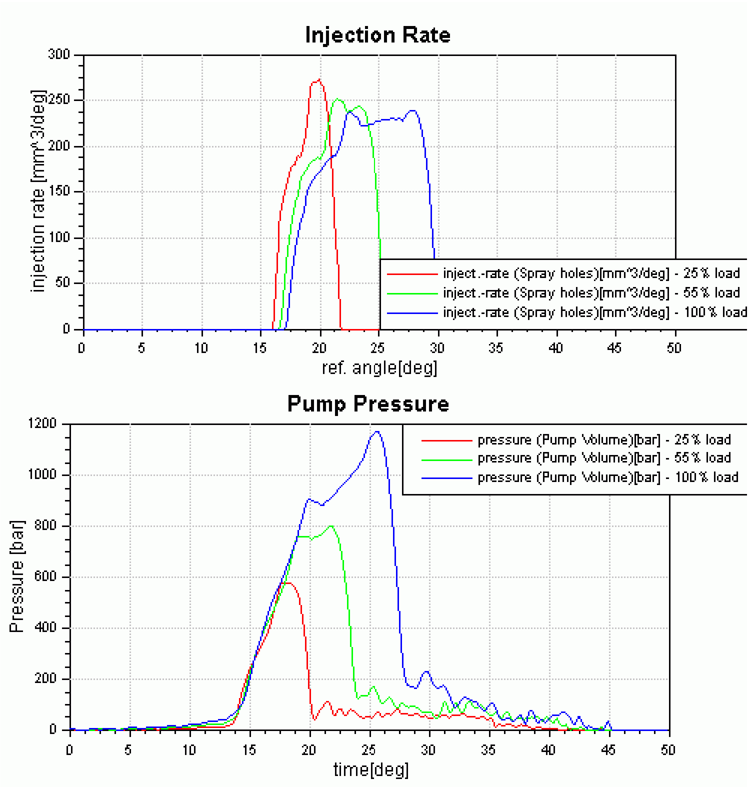


Figure 6-56: Rated Engine Load and Speed Case

6.3.2.2. Change of Injection Line Bore Diameter

In the same manner as in the previous section, create from the original model three Cases for the `hydraulic diameter` of Injection Line2 (No. 9) – `Std_line` (4.5 mm), `4mm_line` (4 mm) and `5mm_line` (5 mm). Store the file under the new name `electronic_pump_line.hyd`.

Figure 6-57 shows the results at rated conditions for the standard injection line, having a 4.5 mm diameter bore and for lines having 4 mm bore and 5 mm bore. All runs were made with the same fuel delivery, using 'Search Adjust' on the solenoid valve closure time T2.

As expected, the injection pressures rise higher and spill occurs earlier, as a smaller bore is used in the line. This is because the hydraulic 'stiffness' of the system increases with reduced line bore. The injection period reduces accordingly.

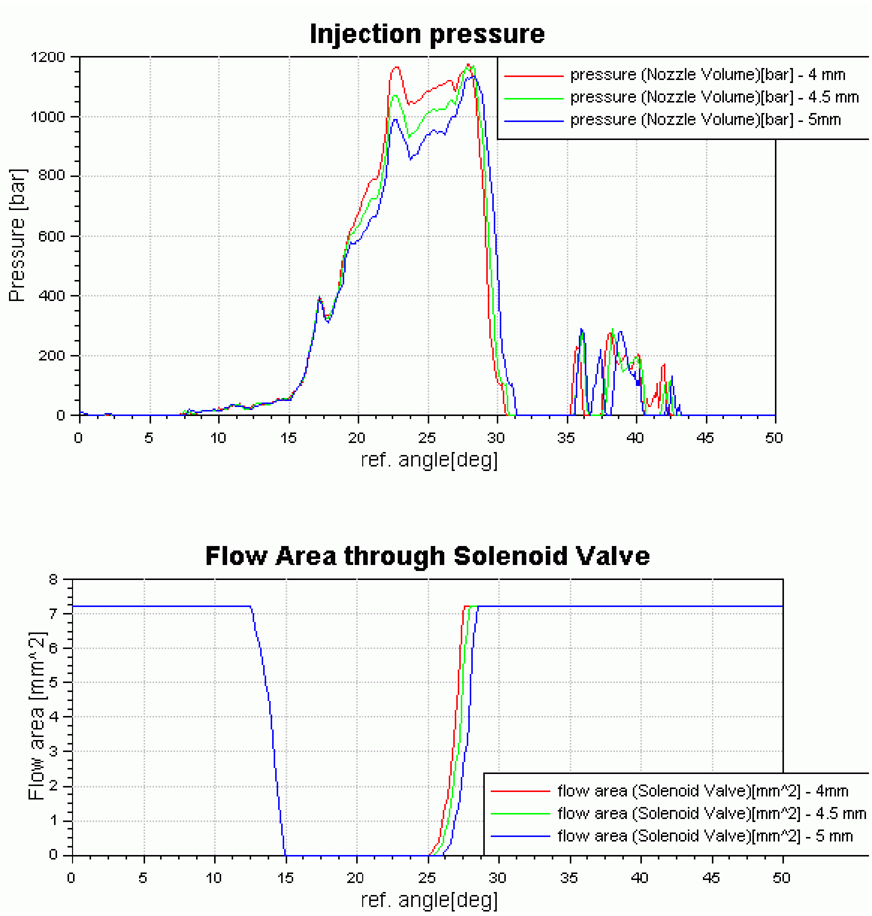


Figure 6-57: Rated Condition for the Standard Injection Line

Note that there is least pressure drop between the pumping chamber and the nozzle gallery using the 4 mm bore line but the pumping pressure is also highest - hence the stress on the fuel cam will also be highest (refer to *Figure 6-58*). The choice of the 4.5 mm bore line is therefore a compromise between minimizing fuel cam / roller follower contact stress and optimizing injection energy at the nozzle.

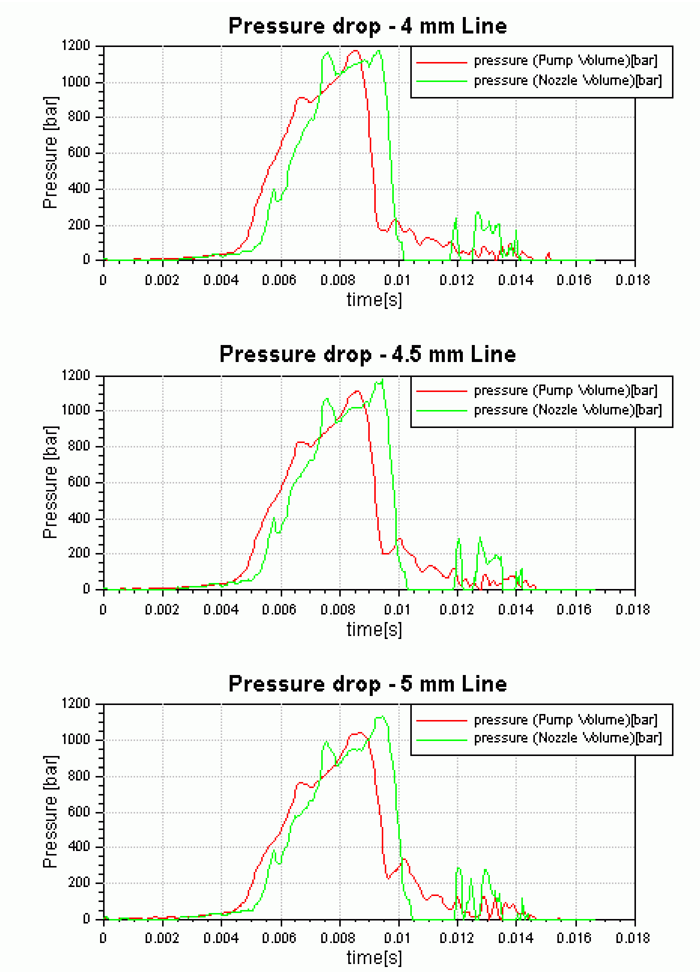


Figure 6-58: Pressure Drop Between Pump and Nozzle Chamber

6.3.2.3. Change of Nozzle Spray-hole Area

As in previous section, create three Cases for nozzles having a 10% increase and a 10% reduction in spray-hole flow area. Store file under `electronic_pump_nozzle.hyd`.

Figure 6-59 shows the results at rated conditions for the standard nozzle spray holes and nozzles having a 10% increase and a 10% reduction in spray-hole flow area. Keeping same injection mass, if we increase nozzle hole area, then we get earlier closing of the solenoid (opening instant is same in all 3 cases). Next, we get lower injection pressure, too, because for higher flow area the flow through nozzle is higher (and vice versa), so the pressure in the nozzle chamber stays lower throughout the injection process.

Comparison of this figure with *Figure 6-57* shows very similar trends - there are small differences around the peak pressure point and the spill characteristics. The final choice of nozzle flow area is determined by the required injection period, allowable peak pumping pressure (pump continuous design limit) and cam/roller follower contact stresses and margin of safety at the cam 'nose'.

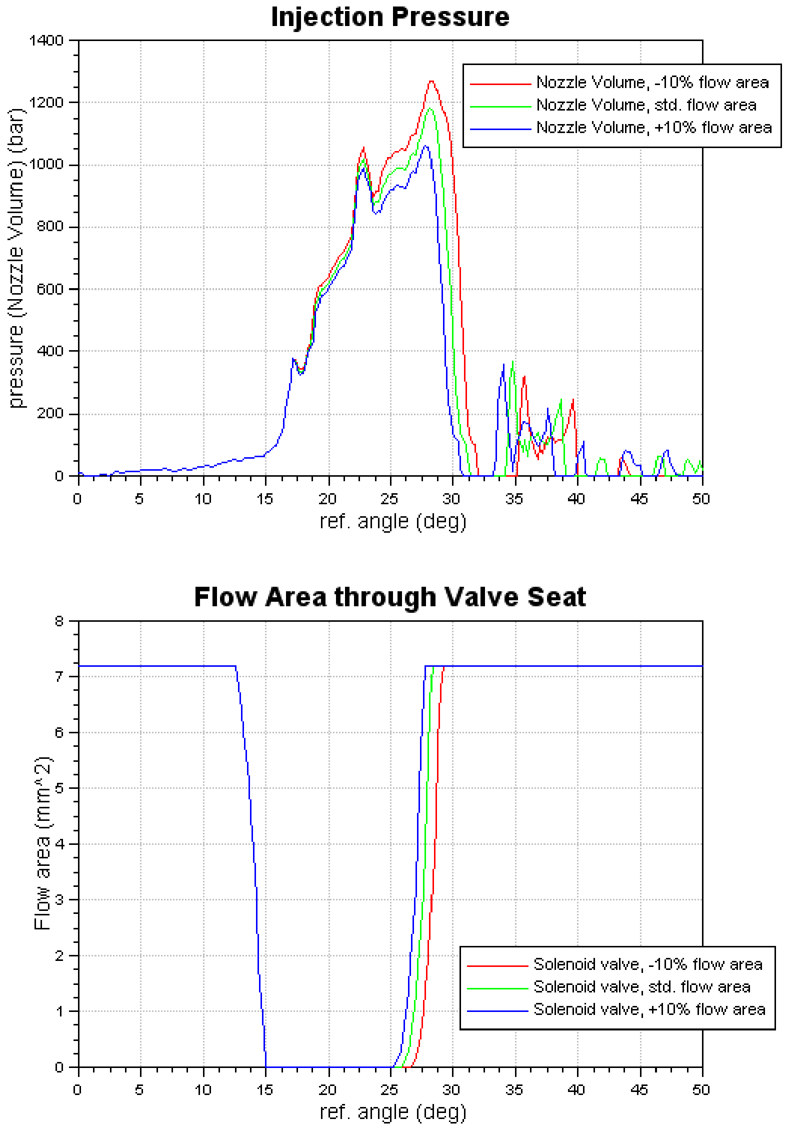


Figure 6-59: Variations of Nozzle Hole Area

7. HYDRAULIC VALVE TRAINS

The use of electro-hydraulic actuation for engine valve trains is becoming more widespread, using electronic control via a fast-acting solenoid or other type of control valve. The use of electronics allows much more flexible valve control for optimal valve performance over the full engine speed range, for such requirements as exhaust gas recirculation for exhaust emissions control.

In this chapter we describe two alternative electro-hydraulically controlled design options, the more complex of which is described in detail and modeled. Such systems can either be used in tandem with the normal mechanical cam-lifted valve operation, for instance for re-opening the engine valve for a short period after closure, or keeping the valve open at the end of its normal open period, or can be totally hydraulically operated (no cam lobe operating).

In the modeling, we assume that a hydraulic oil supply, adequately filtered and temperature-controlled (for protection of the control valve seat and guide surfaces and for acceptable oil viscosity range), is supplied to the engine cylinder head by a 'common rail' feeding all required engine cylinders or valve actuators. The model covers the point of intake of this high-pressure oil supply from the rail, its subsequent supply to the cylinder head drillings via the high-pressure control valve, into the actuator piston/cylinder assembly and spill back to low pressure. Only one engine valve actuator is modeled here. Normally one or two actuators per engine cylinder would be required, if modeling of the full system is to be performed.

AVL BOOST Hydsim now incorporates new, dedicated icons with input dialog capabilities to model several different possible variants of such hydraulic valve actuation mechanisms.

Figure 7-1 shows a typical circuit for such a valve control system.

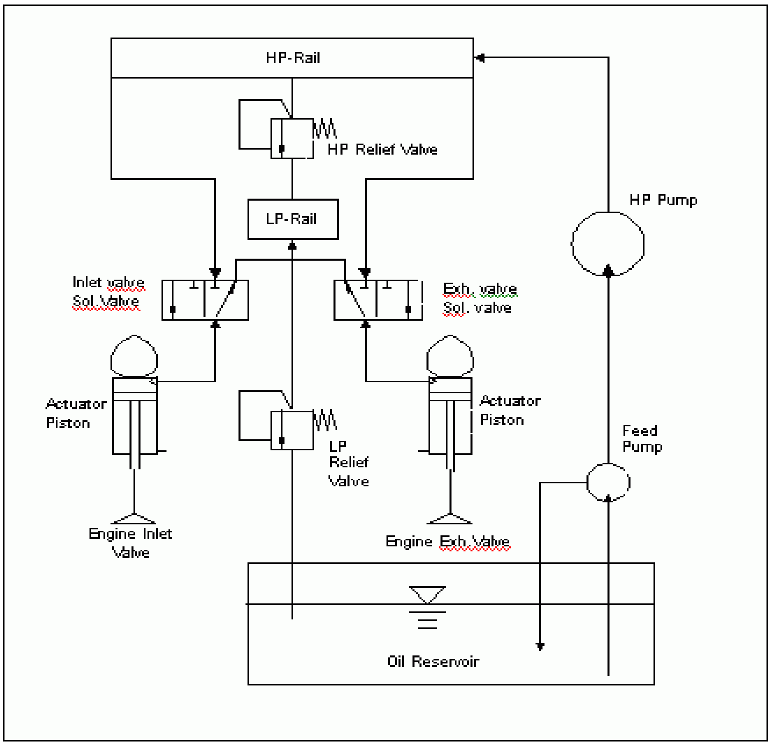


Figure 7-1: Typical Hydraulic Circuit for an Electro-hydraulic Valve Control

7.1. Direct-acting Valve Actuator

Figure 7-2 shows the layout of a direct-acting piston-type hydraulic valve actuator, located between the mechanical valve lifting cam and the engine valve stem. Here, the hydraulic actuation is in-line with the valve stem and no special kinematics need to be taken into account.

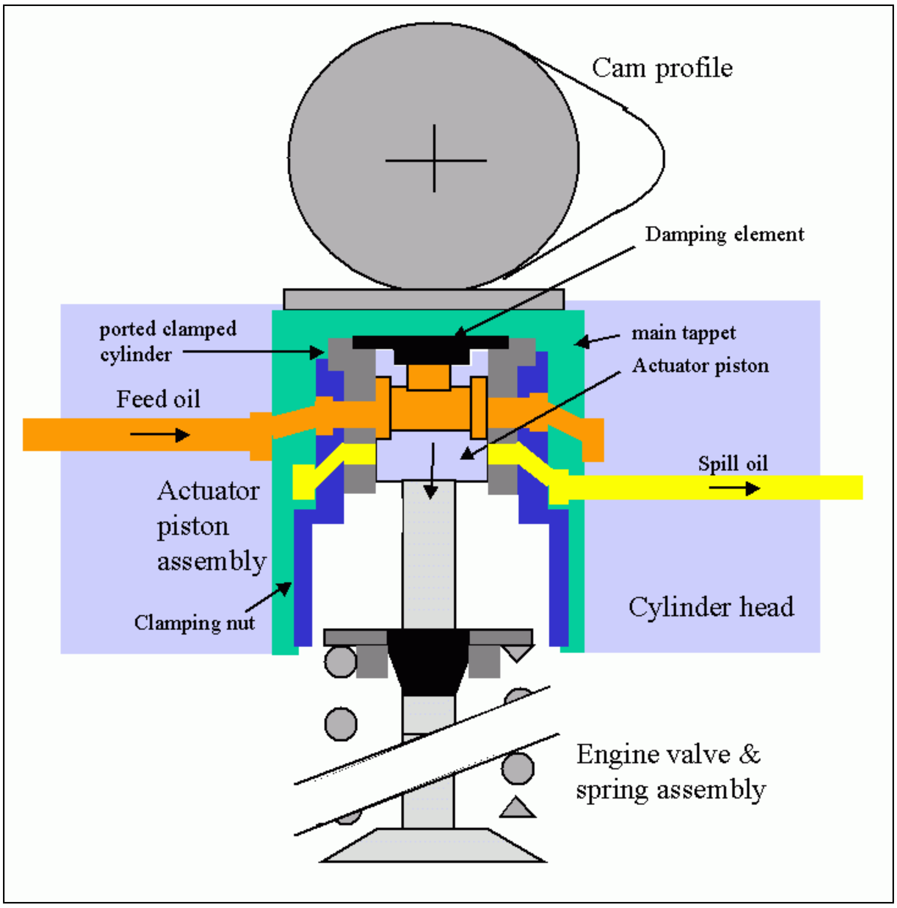


Figure 7-2: Design Layout of Direct-acting Valve Mechanism

In this design, the high-pressure oil supply is fed through drillings in the cylinder head to a groove around the main actuator piston assembly. Oil flows through four angled drillings in the main tappet wall, into a chamber in the clamping nut, which is threaded to the outer tappet and clamps the actuator piston guide, through a flanged 'damping element', to the inner face of the tappet (nearest the camshaft).

The oil then flows through four further holes in the wall of the clamped cylinder, to a groove in the actuator piston outer diameter. The groove feeds the oil via cross-drillings in the piston to the top face of the actuator piston, enabling it to force the piston downward to operate against the engine valve spring and open the engine air valve.

The bores in the wall of the piston guide and the depth of the groove in the piston outer diameter are chosen or developed to produce the required flow control against piston lift, to maintain a stable (fixed height) piston lift, to the required value, under the application of oil pressure. Here, two or more oil feed bores are used (to provide both a reasonably balanced feed around the piston and to give the required flow overlap with the spill port near full lift and to produce the required stable full lift).

Hence, the hydraulic flow through the available area is balanced with the valve spring force, which is trying to re-close the engine valve.

The 'damping element' is a simple stepped spigot, the spigot of which enters the piston top bore, as the piston returns towards its zero-lift (upper) position, at the end of engine valve actuation. The clearance and covering length of the spigot/piston bore diameters determine the level of piston damping before the valve re-seats. This is necessary in order to limit the maximum velocity of the valve at re-seat to a level which will not cause damage.

Whilst this hydraulic model is not shown here, the methods of incorporating these feed and spill bores and the damping element are the same in the detailed model which follows.

7.2. Semi-floating Rocker Mechanism

7.2.1. Hardware Description and Operation

Figure 7-3 shows the layout of a 'semi-floating' design of an electro-hydraulic valve train. In this particular design, an overhead camshaft operates on a semi-floating Finger follower assembly, whose integral roller sits against the cam flank. The hydraulic actuator end of the follower has a hemi-spherical adjuster which sits in a mating cup acted upon directly by the hydraulic piston. The other end of the follower acts directly on the top of the engine valve stem and has also a small, sliding side motion, owing to the follower's rotation as the piston lifts and the fixed ball contact. The BOOST Hydsim program calculates all the kinematics involved, using logical inputs of data from the base hardware into a dedicated element **Finger Follower**. These inputs for the new elements covering the relevant mechanical valve train features are covered in detail later. Note that in the diagram, the engine valve is shown inclined at a small angle, representing a more complex case. Our example uses a valve with no inclination, but the angular input for this is available in the dialog box, if required.

The feed, spill and damping features of the hydraulic actuating piston are also described in detail later.

The high-pressure oil feed (here 100 bar) is made via a 'common rail' and an electronic control valve which supplies a feed bore in the cylinder head to each valve actuator.

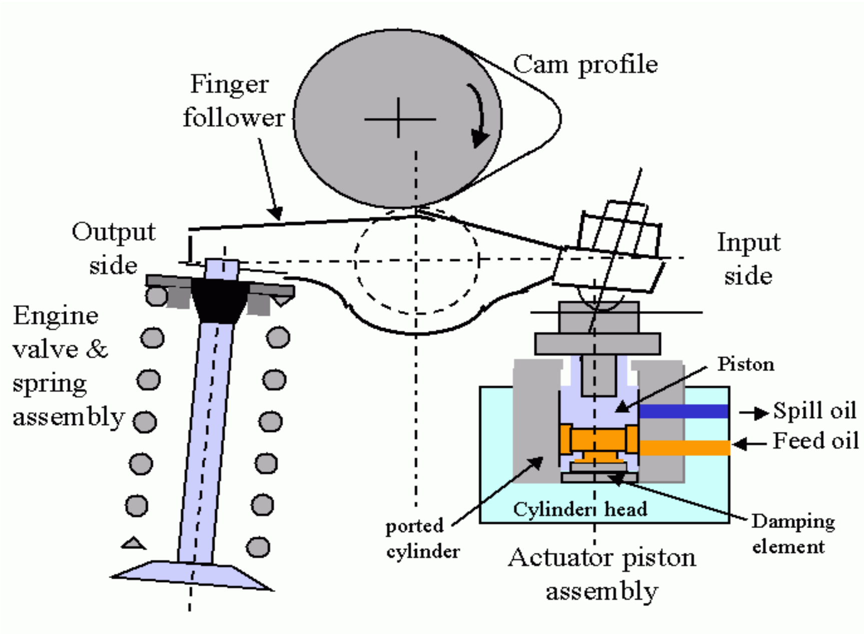


Figure 7-3: Design Layout of a Semi-floating Electro-hydraulic Valve Train

In this design, the actuating rocker, here called a Finger follower has a roller incorporated near to its center, which sits under the overhead cam flank. It has no fixed central fulcrum and can therefore be termed 'semi-floating'. One end of the rocker sits on top of the engine valve stem, able to slide laterally, in line with the rocker longitudinal axis. The other end of the rocker has a screwed, lockable adjuster with a lower spherical ball-end, which sits in a mating spherical cup in the top end of the hydraulically-operated actuator piston. Hence, in this case, at zero actuator lift, the actuator piston is in its lowest, seated position. A clip between a groove in the lower spherical cup and the top of the rocker holds these parts together.

Whilst the ball contact is fixed vertically, the position of the roller main axis, along with the effective radii from the roller to the valve stem contact and the spherical cup contact points will vary slightly, as the roller height changes.

Again, the mechanism may be actuated only by hydraulics (no cam lift designed in) or a combination of hydraulics and the overhead cam. Any required engine valve lift emanating solely from the cam have to be input as a discrete set of acceleration or lift data in the Cam profile element (acceleration data normally produce more accurate results).

BOOST Hydsim automatically takes into account the kinematics of this design but the various required inputs must be input into the **Finger Follower** and other elements. These include all the relevant angles, the lengths from pivots to roller center, masses and finger follower moment of inertia.

7.2.2. Creating the Model

Figure 7-4 shows the BOOST Hydsim model for this system.

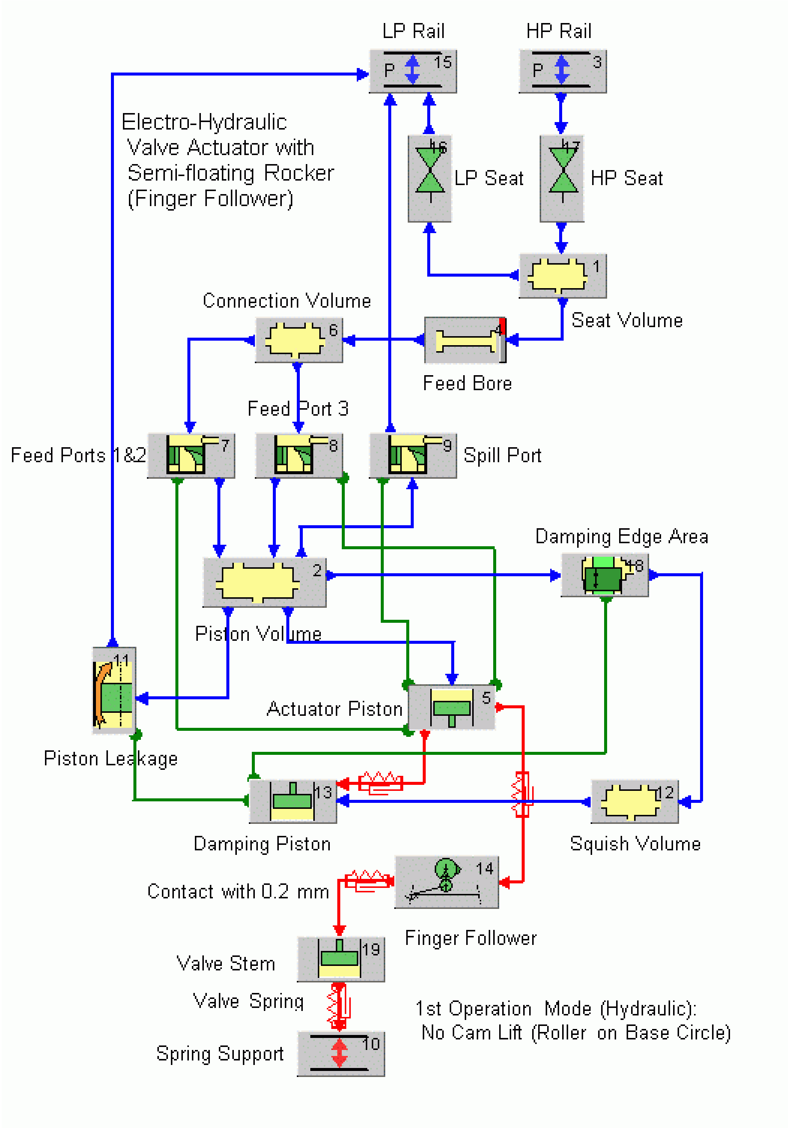


Figure 7-4: BOOST Hydsim Model of Semi-floating Electro-hydraulic Valve Train

The model consists of the following elements:

- HP Rail **(BOUNDARY/Pressure)**
- LP Rail **(BOUNDARY/Pressure)**
- HP Seat **(THROTTLE/Time-controlled switch)**
- LP Seat **(THROTTLE/Time-controlled switch)**
- Seat Volume / Feed gallery **(VOLUME/Standard)**
- Feed Bore **(LINE/Laplace Transform)**
- Connection Volume **(VOLUME/Standard)**
- Feed Ports 1&2 **(PORT/In-line Fill/Spill)**
- Feed Port 3 **(PORT/In-line Fill/Spill)**
- Spill Port **(PORT/In-line Fill/Spill)**

- Actuator Piston **(PISTON/Standard)**
- Damping Piston **(PISTON/Standard)**
- Piston Volume **(VOLUME/Standard)**
- Piston Leakage **(LEAKAGE/Annular gap)**
- Squish Volume **(VOLUME/Standard)**
- Damping Edge Area **(THROTTLE/Lift-controlled)**
- Finger Follower **(LEVER/Finger Follower)**
- Valve Stem **(PISTON/Standard)**
- Spring Support **(BOUNDARY/Mechanical)**

The system begins at the High-pressure Rail 3[7], which is modeled as a Pressure Boundary with a constant pressure of 60 bar. HP Seat 17 connects to the Solenoid Seat Volume 1. The Low Pressure (LP) Seat 16 and High Pressure (HP) Seat 17 have as inputs their effective flow areas and the required start of closure and re-opening for the chosen control valve (giving demanded duration of oil supply). The Solenoid Seat Volume 1 represents all high-pressure trapped volume within the solenoid design. There is normally some overlap between opening and closing of the valve, during which time leakage will occur between the HP rail and the LP rail, through the solenoid valve.

The solenoid control valve feeds the Feed Bore 4. A separate feed to each actuator piston within the cylinder head would normally be required. Here only a single feed and actuator are modeled. A small Connection Volume 6 connects the Feed Bore 4 to the four ports in the guide of the Actuator Piston 5. There are three Feed ports, 7 & 8 in the model and one Spill port, 9. Two of the feed ports have the same diameter and height in the cylinder around the piston, therefore they are represented by one element only (Feed Ports 1&2) in the model. If they had different diameters or heights, a separate element for each would be required.

7.2.2.1. Oil Feed and Spill Port Modeling

In this model, the **In-line / Fill & Spill port** element is used to model the feed and spill bores (as the principles of round port cut-off and re-spill are identical here). The design layout for the feed and spill ports, designed for a fixed 4 mm actuator piston lift, is clearly shown in *Figure 7-5*.

The formulas governing the effective flow area vs. piston lift against feed and spill ports are given in the BOOST Hydsim Users Guide in *Section 13*, *Table 13.2*.

[7] Numbers given for elements may differ from those in your model. Reference *Figure 7-4* for numbered items.

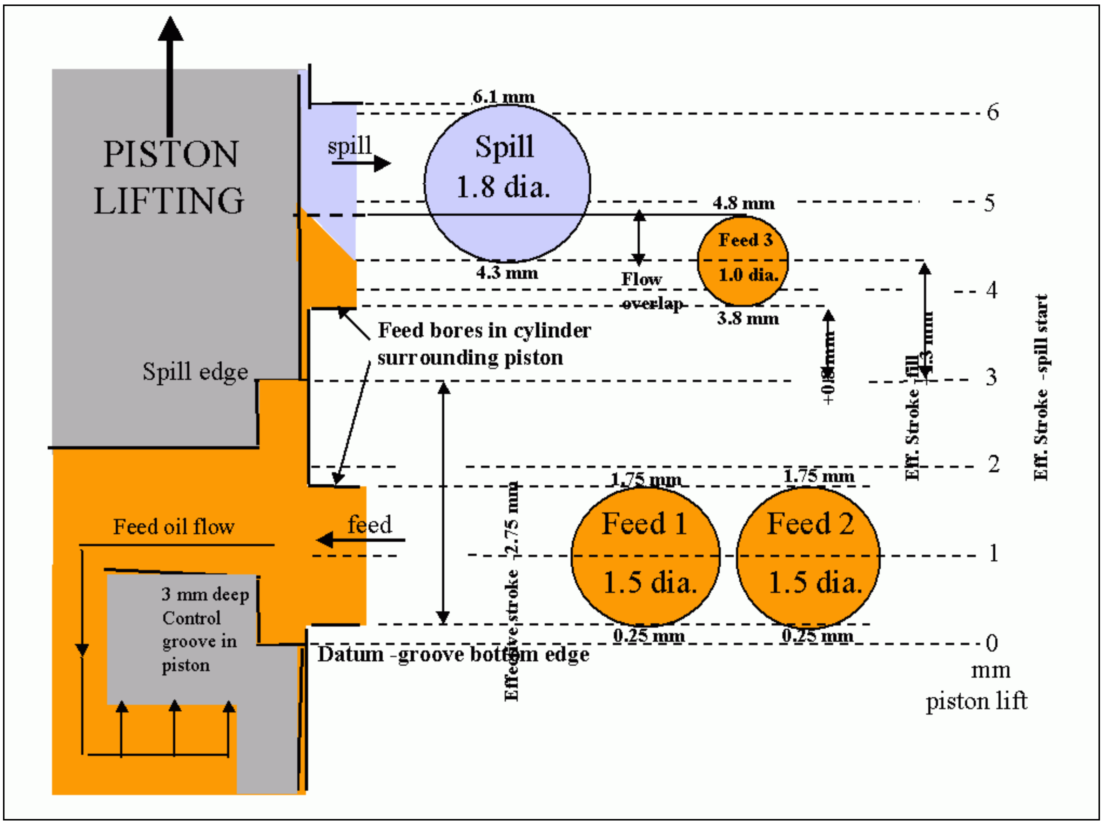


Figure 7-5: Layout of Oil Feed and Spill Ports against the Actuator Piston Groove

A horizontal groove is used to control the flow edges, therefore the 'helix angles' in the input dialog boxes for the ports are all set to zero.

The diameter and 'effective stroke' for each bore has to be input in its dialog box. The 'pre-stroke' is not used.

The effective-stroke for a **feed port** is the distance between the upper (spill) edge of the control groove in the actuator piston and the lower edge of the feed port, i.e. the distance the piston must travel in its normal direction, from its zero lift, seated position, to start flow. Note that in this case the first two feed ports are already open with the piston at its zero lift position, therefore the effective stroke is given a negative value by definition. The upper groove edge must lift 0.8 mm to start to uncover the third feed port, hence it has a positive effective stroke.

The effective stroke for a **spill port** is the distance between the same upper edge of the control groove and the lower edge of the spill port, i.e. the distance the piston must travel in its normal direction, from its zero lift, seated position, to begin to uncover the spill port to spill oil. These dimensions are clearly shown in Figure 7-5.

The user must therefore have a good knowledge of the desired feed and spill port layout relative to the piston control groove. These dimensions are selected normally by initial rough estimation of the required flow areas, then by iteration during detail model optimization and viewing of results, paying attention to the required piston lift and velocity performance.

If an alternative design of feed/spill porting is required, e.g. using rectangular or specially-**shaped ports**, the areas of these against the piston control edge at fixed piston positions must be calculated separately and the data for these then input into the model using general Slide Valve elements (one for combined fill and one for spill port, if both are non-regular shapes).

The 3 mm slot width of the Actuator Piston plays an important role in setting the cut-off and re-spill parameters. The 'narrowest area' is set to a relatively high value (here 10 mm2), since there is no other flow area restrictive to spill other than the ports and groove.

Note also that in this design layout, the upper feed bore (Feed Port 3) and the spill bore overlap in operation towards the required peak piston lift value, to balance the hydraulic force under the piston against the engine valve spring force.

7.2.2.2. Completing the Model

The three feed ports feed the Piston Volume 2, which acts on the Actuator Piston 5. This is the total volume trapped to the side of the piston and within its internal feed drillings. The piston here has a 7 mm guide diameter (a close fit with the piston guide). The user however will note that a second piston element, Damping Piston 13, is connected (via a rigid mechanical connection) to the Actuator Piston 5. This second piston must be formed within the model to define the effective hydraulic areas of the piston ends. The actuator piston is acted upon from its lower end initially only over the area of the damping spigot. As the piston rises to uncover the edge of the damping spigot, it opens up the full diameter to the feed pressure, feeding the Squish Volume 12. This volume is given a low (100 mm3) volume and a low initial pressure (1 bar or atmospheric pressure). The Squish Volume 12 is connected to the Piston Volume 2 via Damping Edge Area 18 (**Lift-controlled throttle**). Input for this element is the initial annular leakage area between the spigot and piston bore and the effective area vs. piston lift.

The Actuator Piston 5 is connected to the Finger Follower 14 (element of **Lever** group) using a mechanical connection, which is given no pre-load but a relatively high stiffness and damping value (fixed contact between piston and cam side lever arm). The Finger Follower 14 is connected to the engine Valve Stem 19 by a ‘variable’ mechanical connection, which represents the contact stiffness between valve side lever arm and valve stem (with 0.2 mm clearance). The Valve Stem 19 is modeled as a **Standard Piston** element. Only the valve mass, including one third of the spring, need be entered here. The valve spring preload, rate and a suitable damping value are input into a mechanical connection between the Valve Stem 19 and the Spring Support 10 (mechanical boundary).

Valve body positive motion (see Figure 7-8) is opposite to positive motion in global coordinate system and because of that for valve stem and its mechanical connections negative direction of motion must be defined.

The leakage past the piston/cylinder clearance is input as the Piston Leakage 11 between the Piston Volume 2 and the Low Pressure rail 15 (set here at 1 bar, atmospheric pressure). Note that there is also loss of high-pressure oil to the LP rail 15 during switching of the solenoid valve in either direction. The Spill Port 9 in the actuation cylinder feeds directly to the LP rail, too.

7.2.2.3. Finger Follower

One new element, a semi-floating rocker or **Finger follower** is required to model this valve control system, or others of this type. The main input dialog of Finger Follower is shown in *Figure 7-6*.

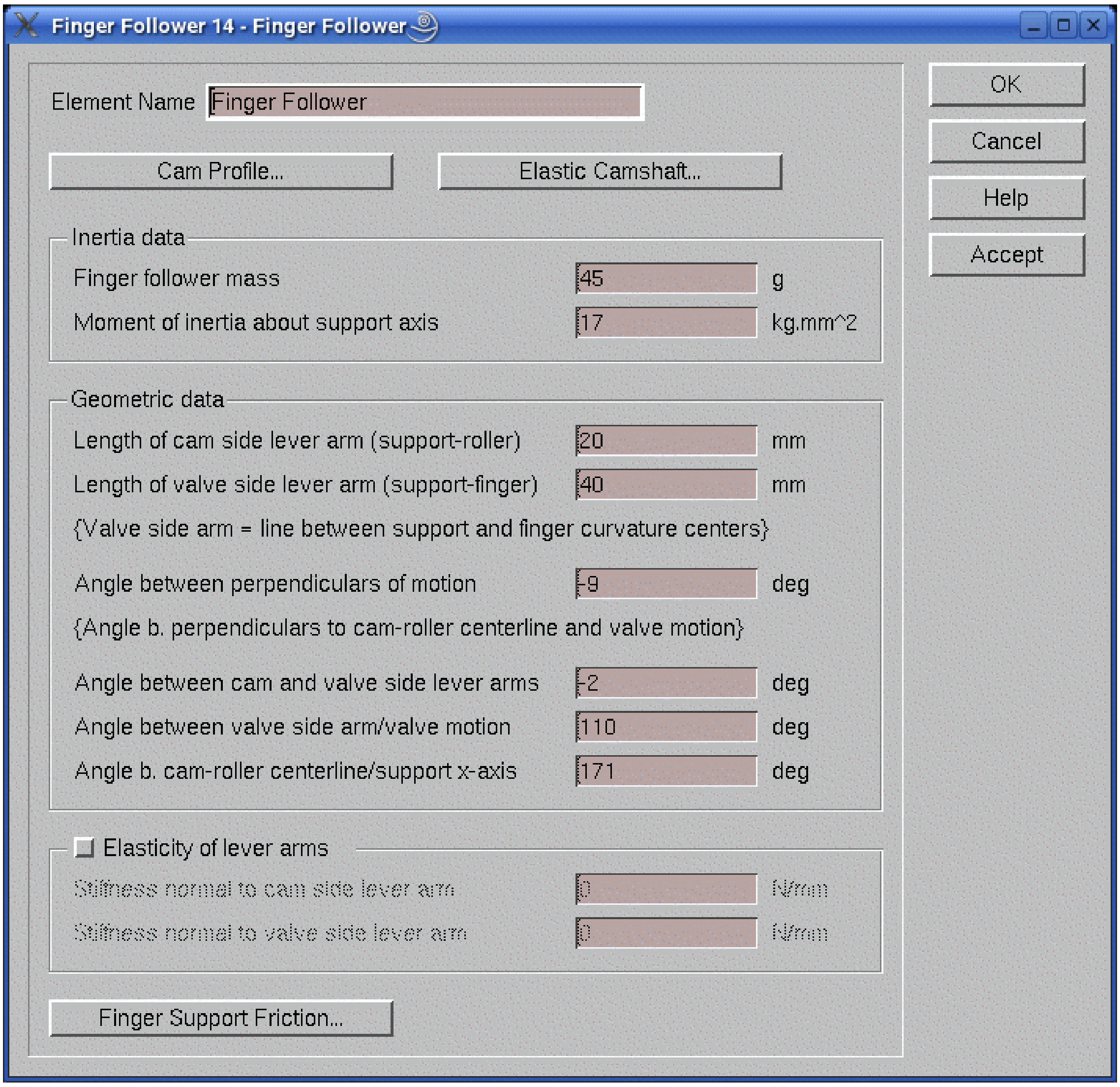


Figure 7-6: Finger Follower Input Dialog

Note that the following sub-dialog boxes are available within this Finger Follower for the cam and roller follower inputs:

- **Cam Profile**, into which cam and roller features must be input, including the direction of rotation of the cam.
- **Elastic Camshaft**, into which features for the elasticity of the camshaft bearing support and its torsional stiffness, inertia and damping can be input
- **Finger Support Friction**, into which frictional parameters of finger support can be input

The Cam Profile sub-dialog is shown in Figure 7-7. It must be activated and filled for any cam-operated system to be effective. The other two sub-dialogs do not need to be activated, depending on the user requirements.

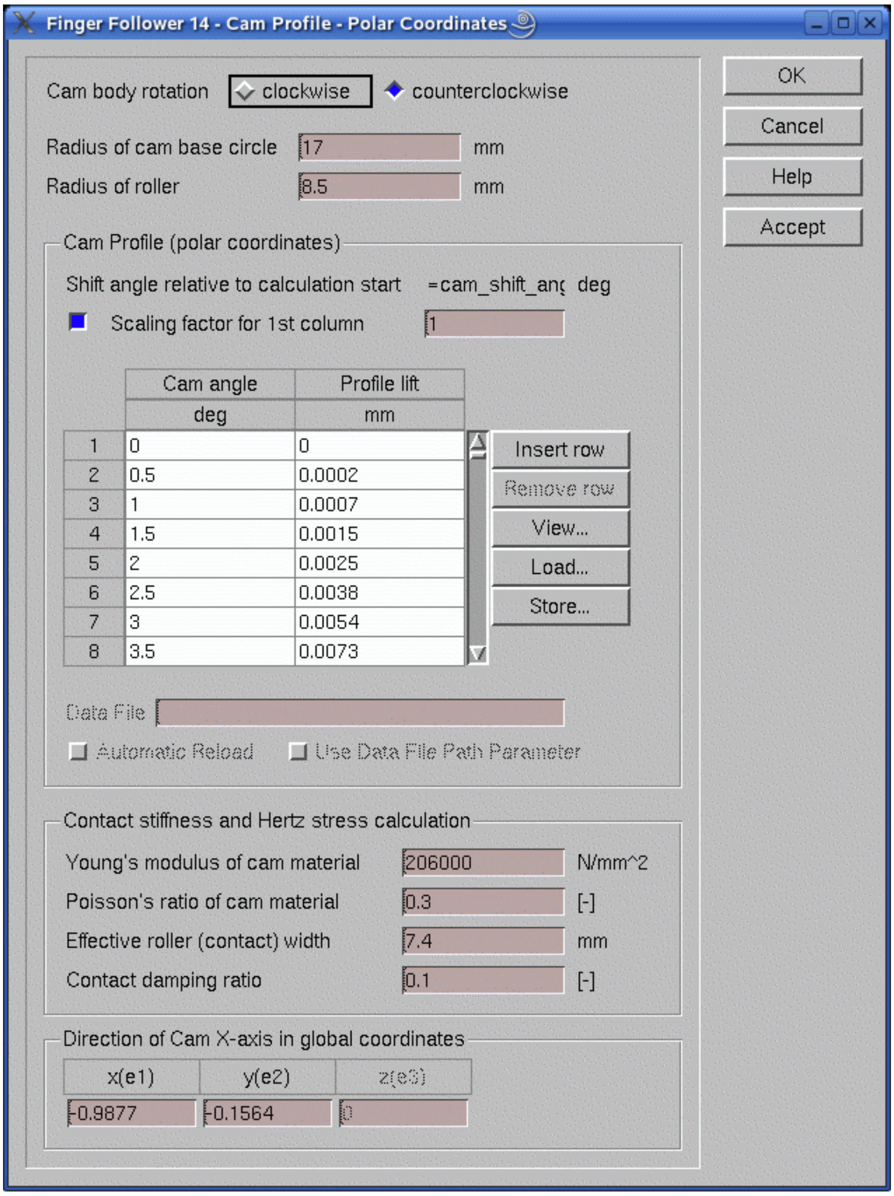


Figure 7-7: Cam Profile Input Dialog

The cam profile inputs include the cam base circle radius, roller radius, the polar coordinates for the cam (profile lift is the rise in the cam profile from its base circle at a given cam angle). The cam/roller contact stiffness and Hertz stress are calculated by inputting the Young's modulus and Poisson's ratio for the cam material and the effective contact width of the roller. It is necessary for calculation of finger follower and camshaft dynamics.

The Finger Follower element can be given initial conditions via **Element | Initial Conditions**, when the element is highlighted, if the calculation starts from a non-zero lift position.

In the main dialog box, the kinematics features of the valve-train assembly are input. These comprise two lengths and four angles. Detail description of them will be given in further text.

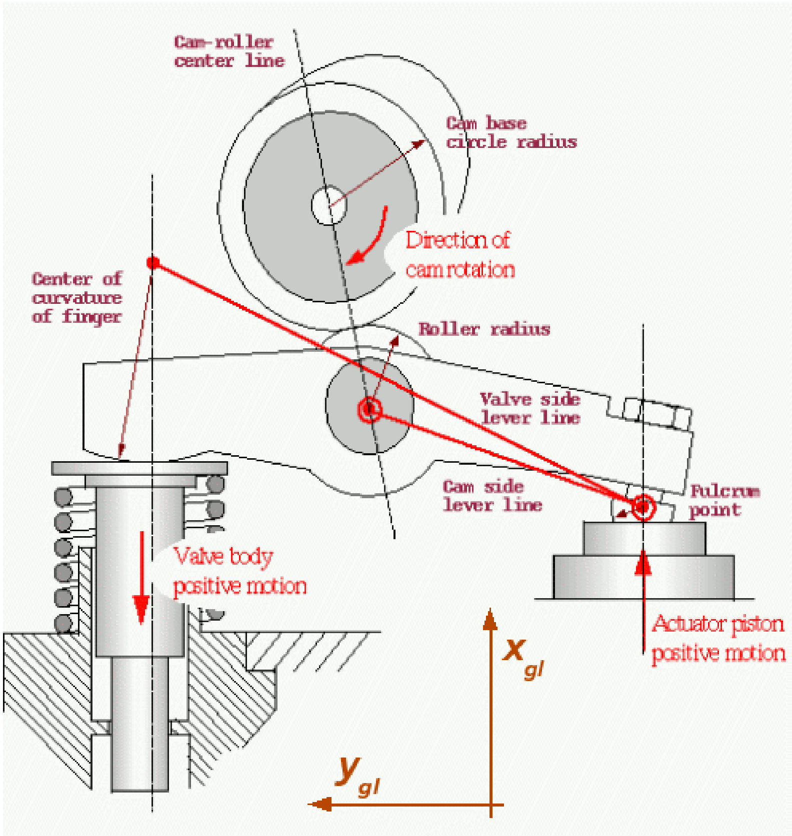


Figure 7-8: Finger Follower Schematic

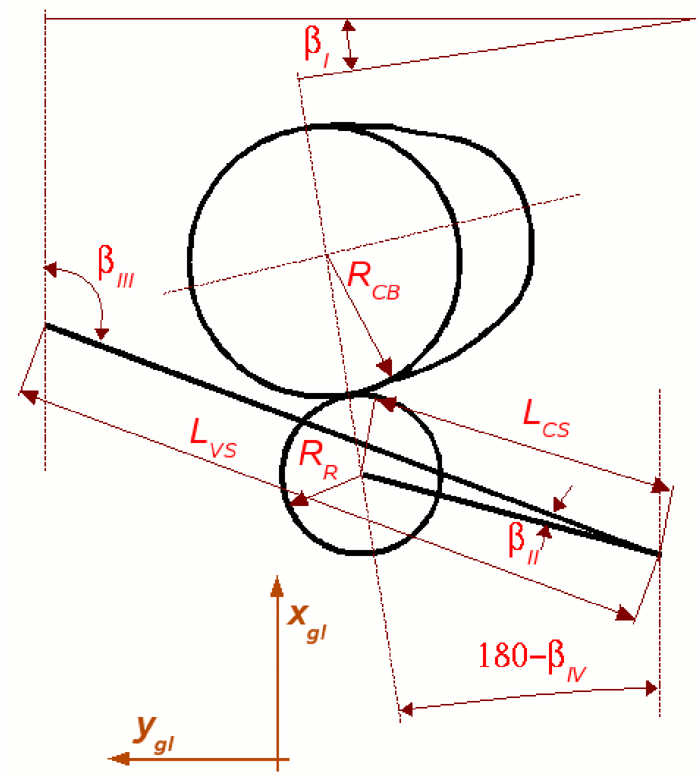


Figure 7-9: Kinematics Diagram Showing Required Length & Angle Inputs

A description of the features in this Finger follower dialog box is given below:

- The total mass of the follower or rocker assembly (arm, roller, roller bush, adjuster and lock-nut) and its moment of inertia around the finger follower center of rotation axis (fulcrum point).
- Length of cam side lever arm (support-roller). This is the length of the arm from the input fulcrum point to the roller center ('support' here is the spherical contact point above the actuator piston and relates to the 'input' side of the follower). Length L_{CS} in Figure 7-9.
- Length of valve side lever arm (support-finger). This is the length of the arm connecting the input fulcrum point, to the center of curvature of the finger, which pushes the valve stem. Length L_{VS} in Figure 7-9.
- Angle between perpendiculars of motion. This is the angle included between a line drawn perpendicular to the line joining the cam and roller centers and a line drawn perpendicular to the center-line through the engine valve. This is angle β_I in Figure 7-9 and here it is a negative (see conventions in the GUI Users Guide Chapter 5.3.1).
- Angle between cam and valve side lever arms. This is the angle included between the line drawn between the actuator input spherical center and the roller center (cam side lever line) and the line drawn between the same input actuation point and the center of curvature of the valve finger radius (valve side lever line). This is angle β_{II} in Figure 7-9 and here it is a negative (see conventions in Reference Manual Chapter 5.3.1).
- Angle between valve side arm/valve motion. This is the angle between the line joining the center of curvature of the finger and the actuator spherical center and the negative axis of the engine valve. This is angle β_{III} in Figure 7-9 and here it is a positive (see conventions in the GUI Users Guide Chapter 5.3.1).
- Angle between roller-cam centerline and support negative x-axis. This is the angle between the line joining the center of roller and the center of cam base circle and the negative axis of the engine valve. This is angle β_{IV} in Figure 7-9 and here it is a positive (see conventions in the GUI Users Guide Chapter 5.3.1).

Cam rotational direction

The direction of rotation of the cam has an effect on the kinematic motion of the roller, as the cam center is offset to the roller vertically. This is accounted for in the model by specifying the direction of cam rotation and activating one of the toggle buttons in the Cam Profile sub-dialog box.

7.2.3. Running the Model and Typical Results

The model is here run in two modes.

The first mode assumes no camshaft input to the motion of the finger follower (no cam lift flank operative, i.e. a fully-hydraulic finger follower operation and effectively a 'cam-less engine'. This is made possible by inputting a large 'Shift angle relative to calculation start' in the Cam profile sub-dialog box of Finger follower, effectively telling the program that the hydraulic actuation will be completed well before the cam starts to lift the roller.

The second mode assumes camshaft lift of the roller to full lift, return of the roller back down the other side of the cam to a point around 2.5 mm lift of the engine valve (its maximum lift is around 10 mm). The hydraulic actuation is then implemented to keep the engine valve depressed, as the cam lift further reduces and the roller follower returns back further towards its zero lift position. This 'catches' the engine valve at around 2.5 mm lift and keeps it open for a demanded extra period of time, for say engine emissions control management.

7.2.3.1. First Mode of Operation – No Cam Operative

The 'Shift angle relative to calculation start' is set here formally to 359 degrees. Hence, the cam will not start to lift the roller until 359 degrees cam from the start of calculation of the hydraulic valve operation.

In the BOOST Hydsim model (Figure 7-4), we assigned the following parameters **Model | Parameters** (Figure 7-10):

- Cam Shift Angle Interval (here 359 degrees reference angle)
- Reference Angle Interval (here 170 degrees reference angle)
- storing output values (here 2 x Reference Angle Interval, i.e. two values per cam angle)
- crank angles at which the solenoid starts to close the (initially open) connection to the low pressure rail and to open the (initially closed) connection to the high pressure rail, i.e. to start the hydraulic actuation event
- crank angles at which the solenoid starts to re-open the (closed) connection to the low pressure rail and to re-close the (open) connection to the high pressure rail, i.e. to finish the hydraulic actuation event. For this first run, we select a pressurizing angle of 100 degrees crank.

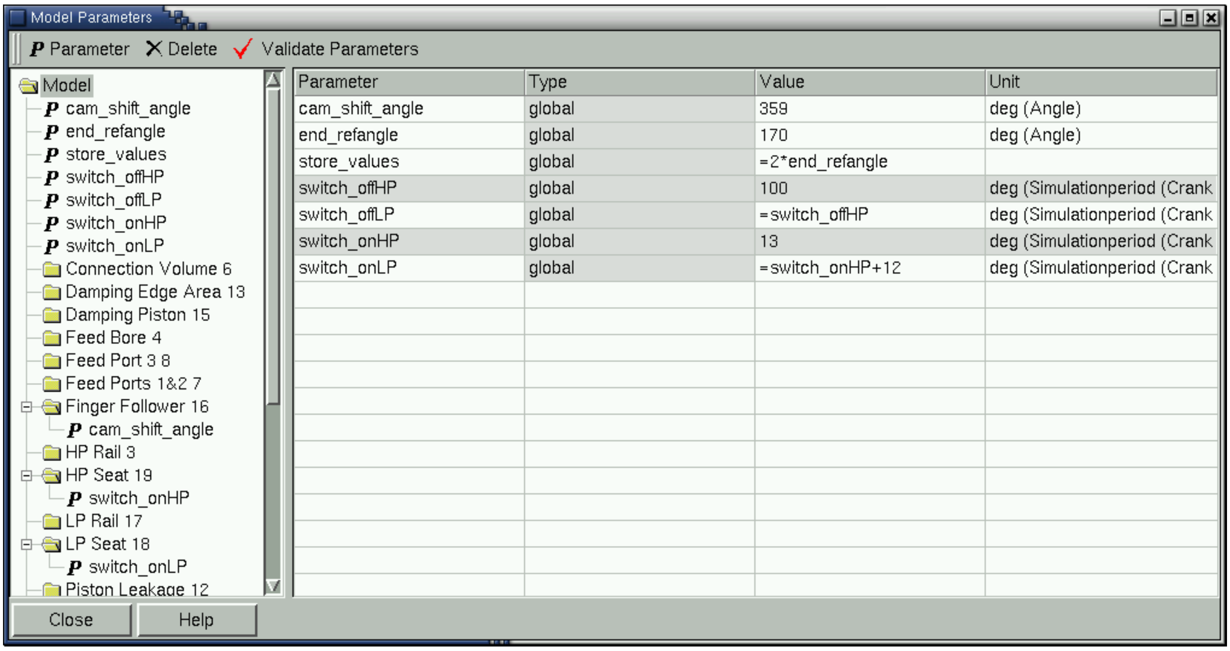


Figure 7-10: Model Parameters Dialog Box

Note that two parameters are grayed out. These parameters are used in the **Model | Case explorer** (*Figure 7-11*).

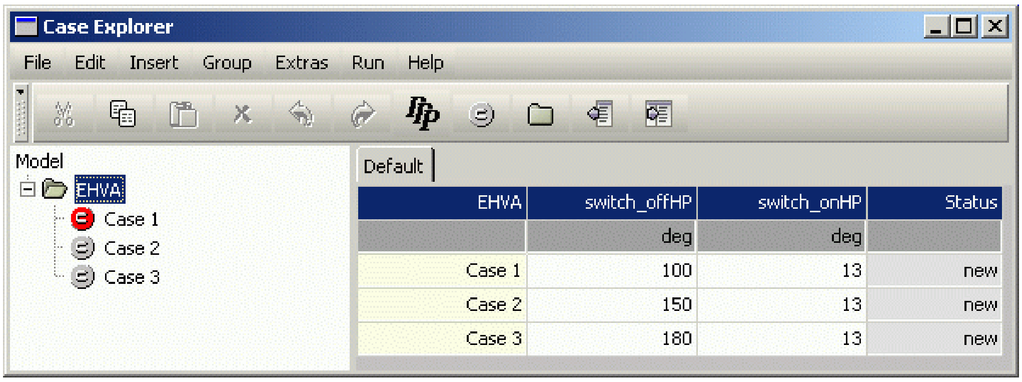


Figure 7-11: Case Explorer Dialog

We then select all useful results to be stored, opening each element in the model in turn. Stored results include:

- Effective flow areas of High and Low pressure seats
- Pressures in certain volumes, including the actuator piston volume
- Instantaneous and cumulative flow-rates through feed and spill ports
- Actuator piston lift and velocity
- Engine valve stem lift and velocity
- Instantaneous and cumulative leakage past the piston
- Cam follower lift etc.

Note that the new Finger follower element has a large number of possibilities for results storage and later retrieval.

To run these selections, select **Simulation | Run**, then highlight all three Cases made, then select **Run.**

Figure 7-12 shows the main results for the three different demands for these runs.

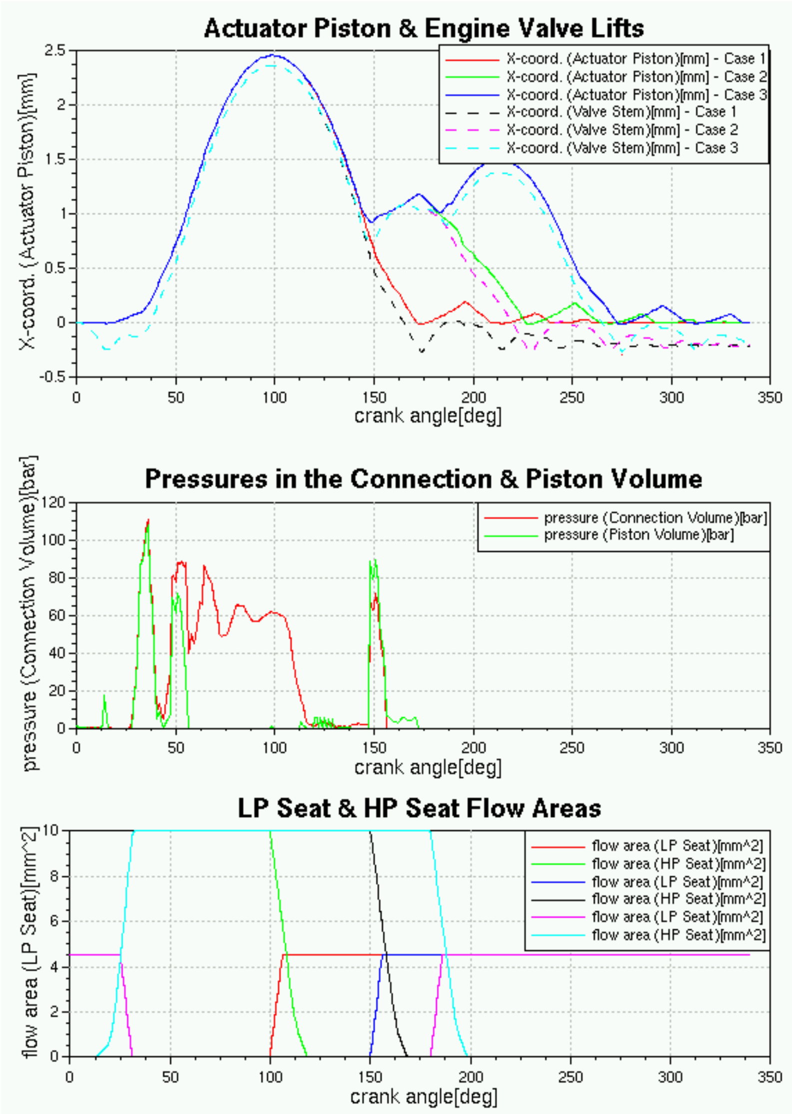


Figure 7-12: Actuator Piston & Engine Valve Lifts & Control Flow Areas in Piston Cylinder for Three Demand Cases

In the top layer:

- Actuator piston lift (solid lines)
- resulting engine valve lift (broken lines), transferred through the finger follower at the rocker ratio input into the Finger follower dialog

In the second layer:

- pressures created in the connection volume feeding the feed ports and under the hydraulic piston

In the third layer:

- LP Seat and HP Seat flow areas, showing the three selected actuation times for the hydraulic feed.

Only the top and third layers show results the three different demand periods, to keep the trace simple.

The results show that the piston lift is dependent very much on the uncovered feed area and the relationship between the feed bores. The selected diameters, heights in the actuation cylinder and number of bores, also the overlap of the upper feed bore 4 with the top spill bore, all have an effect on the lift features of the piston. A balance between hydraulic feed force/area and mechanical spring force on the engine valve is aimed for, to maintain a fixed piston lift. An initial peak is formed, followed by lower piston lifts, as the solenoid control valve duration is increased.

The 'squish effect' to damp the piston as it returns to its seat at the end of lift can be seen, as a slightly reduced piston return velocity.

7.2.3.2. Second Mode of Operation – Engine Cam Initially Operative, followed by a superimposed hydraulic piston lift

We are here interested in the engine valve cam return section, close to and above 2 mm valve lift, so a different 'Shift angle relative to calculation start' has to be used, to set the cam lift correctly against the piston lift. Here, we have set this angle to 0 degrees crank (no shift) in the **Finger follower | Cam profile** sub-dialog box.

We also have to set much later crank angle timing for the hydraulic operation, by putting new timing in the **Model | Case explorer** dialog box. We have used 260, 310 and 360 degrees crank for switching off the high pressure oil supply, respectively. Switching on the high pressure supply is set to 173 degrees crank and is constant for all cases.

Figure 7-13 shows the results, built in a similar manner in the Impress Chart window as before.

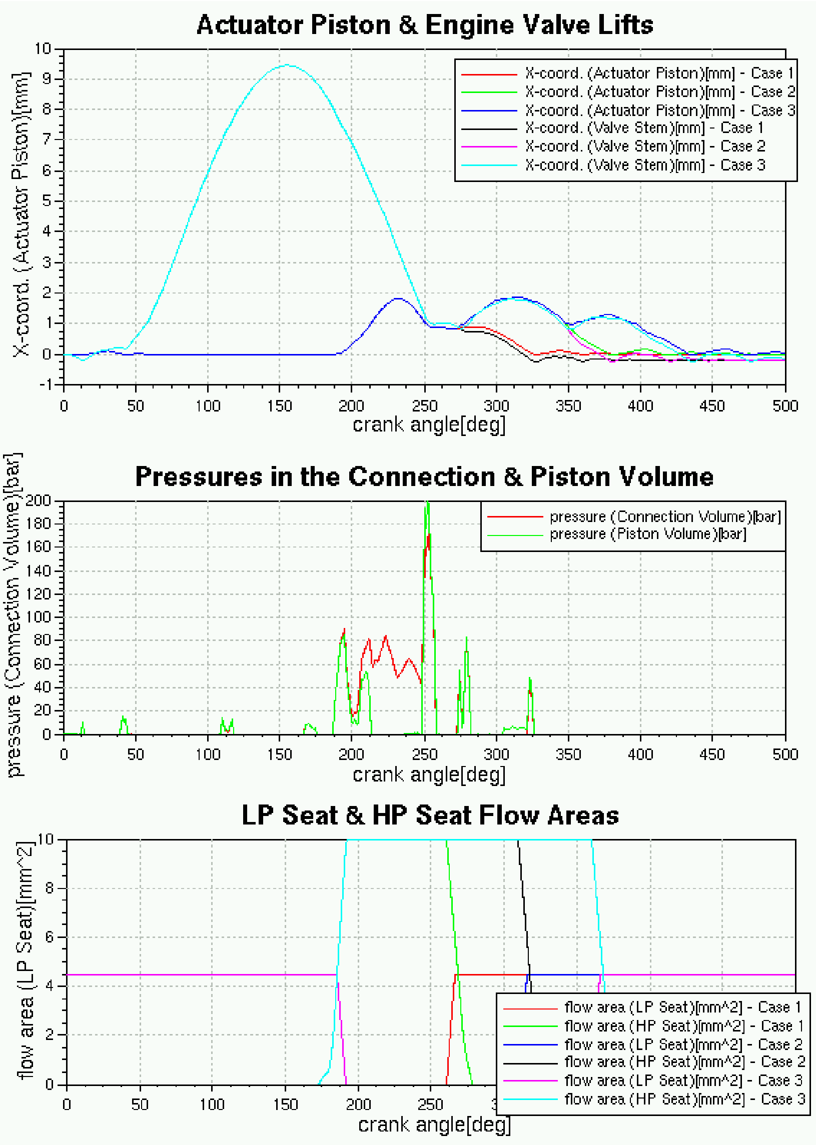


Figure 7-13: Results for Combined Cam and Hydraulic Operation of Finger Follower/Engine Valve

Similar features are displayed in the top three layers.

In the top Impress Chart result layer the engine Valve Stem lift is shown. This has an initial rise to ca. 9.5 mm, from the cam actuation. As the roller reaches a point ca. 2 mm valve stem lift, the hydraulic actuation comes into effect, starting to lift the actuator piston to 'take up the slack' against the mechanical lift by the cam. At this lift the hydraulic force balances the force of the valve return spring and keeps the valve stem lifted for a further period, as demanded by the closure time of the high-pressure oil feed. The upper layer shows the effects of two further oil feed periods. Note that the duration of these two further runs are the same as the first but the hydraulic actuation is shifted later in the cam operation cycle.

The second layer shows the pressures created in the connection volume feeding the feed ports and in the volume under the actuator piston, for the first run only.

The third layer shows the high-pressure and low-pressure solenoid flow areas, indicating the start and end of the three selected pressurization periods.

8. BOOST HYDSIM – MATLAB® INTERFACE

This chapter guides the user through the modeling of BOOST Hydsim -MATLAB® Interface, which is accessible through MATLAB® elements from the Element tree.

There are three options to incorporate MATLAB® model into a BOOST Hydsim model:

- MATLAB® API : Simulink model
- MATLAB® API : m-function
- MATLAB® DLL model

To call MATLAB® `PATH` from BOOST Hydsim on Linux platforms, the user has to define the location of the API shared libraries and set the path of executable `matlab.exe`. It has to be defined in the shell where BOOST Hydsim is started.

- In C shell, the command to set the library path is:

```
setenv LD_LIBRARY_PATH=<MATLAB>/extern/lib/$Arch:$LD_LIBRARY_PATH
```

- In C shell, the command to set MATLAB® path is:

```
set path = (<MATLAB>/bin $path)
```

- In Bourne or bash shell, the commands to set the library path are:

```
LD_LIBRARY_PATH=<MATLAB>/extern/lib/$Arch:LD_LIBRARY
export LD_LIBRARY_PATH
```

- In Bourne or bash shell, the command to set MATLAB® path is:

```
PATH=<MATLAB>/bin:PATH
export PATH
```

where `<MATLAB>` is the MATLAB® root directory and `$Arch` is your system architecture.

The environment variable name `LD_LIBRARY_PATH` may vary on different platforms.

It is convenient to place the above commands in a startup script such as `~/.cshrc` for C shell or `~/.profile` for Bourne/bash shell.

8.1. Model with MATLAB® Simulink

In *Chapter 5.1*, a model of the common rail injector with a two-way solenoid valve is shown. This model will be used as a basis for creating a new model with MATLAB® elements which are aimed to model the pressure control in the common rail. For this purpose, two new Volume elements - Pump and Rail Volume - and two additional Switch valves – Fill and Spill Valves – are added to the system. These valves are controlled by PID Controllers represented by MATLAB® elements.

The system layout is shown in *Figure 8-1*. Compared to the common rail injector model in *Section 5.1*, the following elements are added: Pump Flow (No. 19[8]), Pump Volume (No. 23), Rail Volume (No. 20), Fill Valve (No. 25), Spill Valve (No. 26), Spill Pressure (No. 13), Over-pressure Valve (No. 24) and two MATLAB® elements: Fill Controller (No. 21) and Spill Controller (No. 22). The principle of the control system operation is based on the opening and closing of Fill and/or Spill Valves according to the difference between the target pressure (1500 bar) and actual pressure in Rail Volume. If pressure in the Rail gets above target pressure, Spill Controller has to open Spill Valve and, with Fill Valve closed, the pressure in the rail would decrease. In the opposite case Fill Controller has to open Fill Valve and pressure in the rail would increase, assuming that pressure in Pump Volume is high enough to feed the rail. Both Valves have opening and closing times as well as opened and closed delay time.

For safety, an Over-pressure Valve with the opening pressure of 1800 bar is included into the system. It should never open at normal system operation.

All elements belonging to the control system are connected via wire connections () with appropriate MATLAB® elements. It is necessary for the data exchange through the input and output channels of MATLAB® elements.

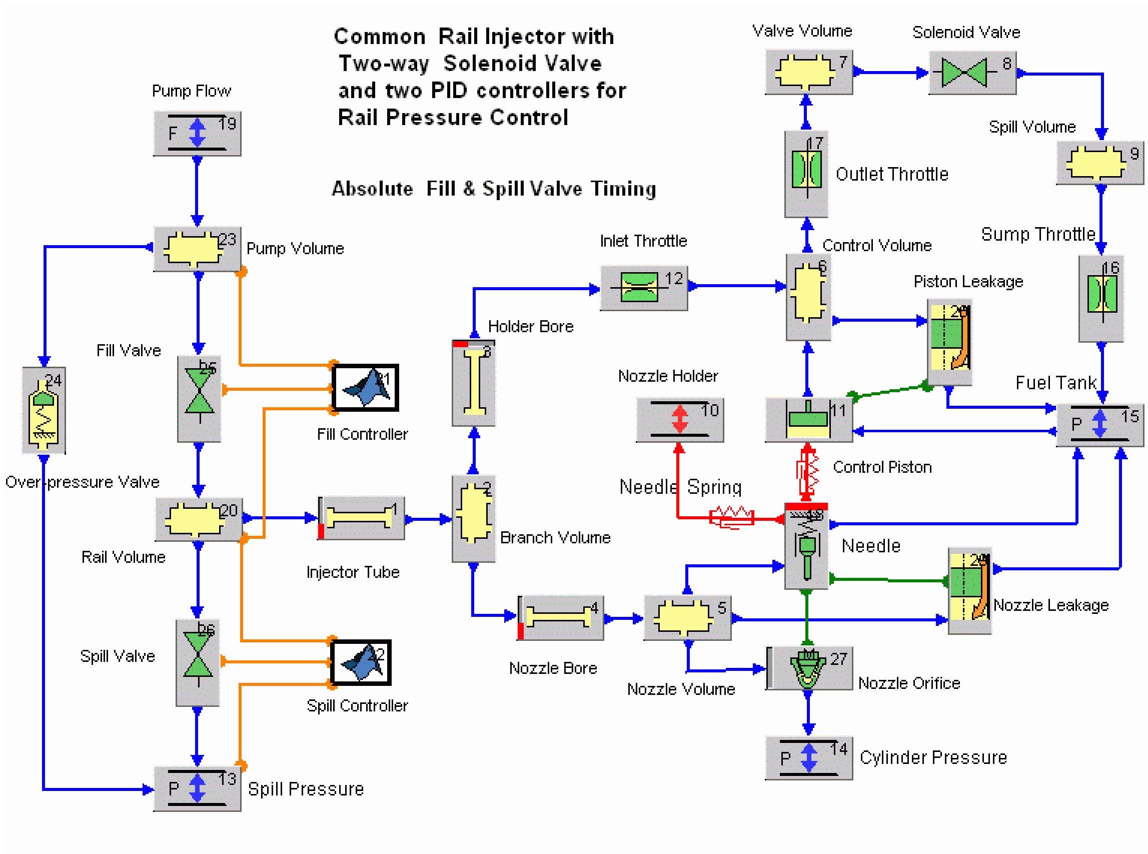


Figure 8-1: BOOST Hydsim Model of Common Rail Injector with MATLAB® Elements as PID Controllers

[8] Numbers given for elements may differ from those in your model. Reference Figure 8-1 for numbered items.

8.1.1. Input Data

In this section only the input dialogs of Rail Volume, Spill/Fill Valves and Fill/Spill Controllers will be shown. Other elements have the same input data as the previous model from *Section 5.1*. Additionally, parameters in the **Calculation Control** input dialog - **Calculation end time, Time step and Number of values to store** – are changed. These parameters are defined as global variables and set to 0.015 s, 3e-7 s and 500, respectively.

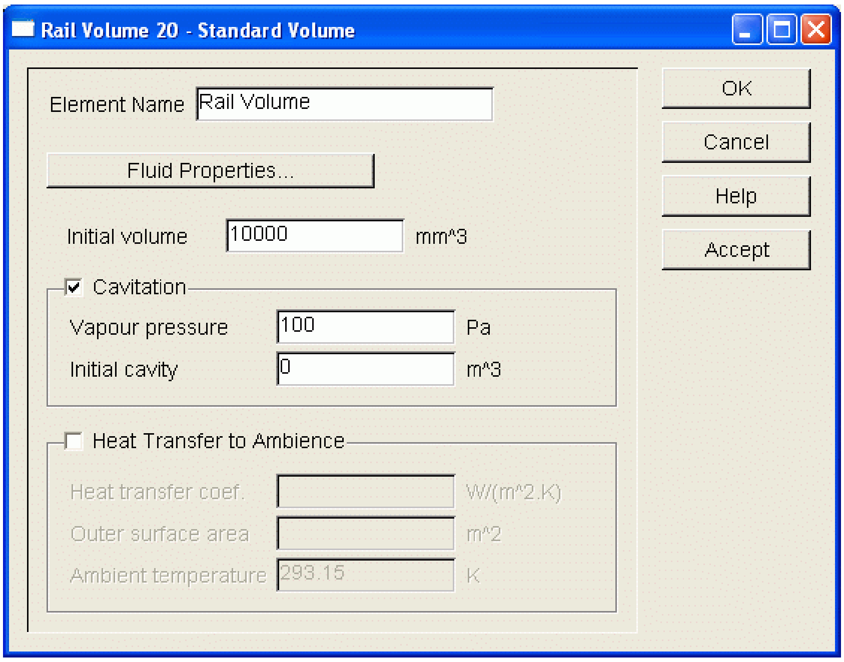


Figure 8-2: Input Dialog of Rail Volume

The Rail Volume size is set to 10000 mm^3. Pump Volume has 10% of Rail Volume size, i.e. 1000 mm^3. Initial pressure in all high-pressure volumes is equal to target rail pressure, in our case 1500 bar.

The input dialogs of Fill and Spill Valves are shown in Figure 8-3 and Figure 8-4, respectively. Opening and closing time instant of Fill/Spill Valves are defined in absolute time domain. Here they are formally set to dummy values (0.01 and 0.003 s) because PID Controllers will redefine them during BOOST Hydsim-MATLAB® co-simulation.

Note: As the interface between BOOST Hydsim and MATLAB® starts from second calculation step (when BOOST Hydsim calls MATLAB® first time), these dummy values have to be greater than second calculation time, i.e. greater than double time step from **Calculation Control**.

Opening/closing times and flow areas of the valves are assigned to global variables (parameters) which are defined in **Parameters** dialog:

Parameter name	Parameter value (m2)
miu_A_fill	1e-6
miu_A_Spill	6e-9

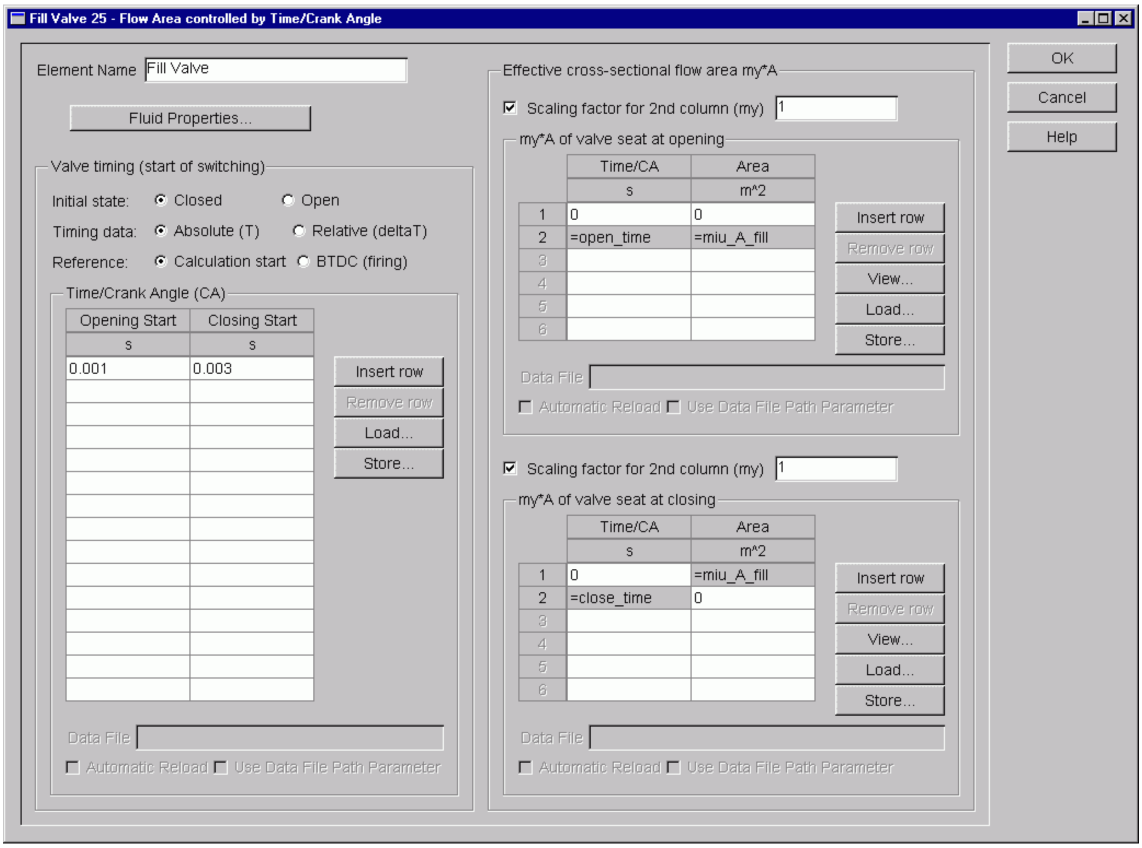


Figure 8-3: Input Dialog of Fill Valve

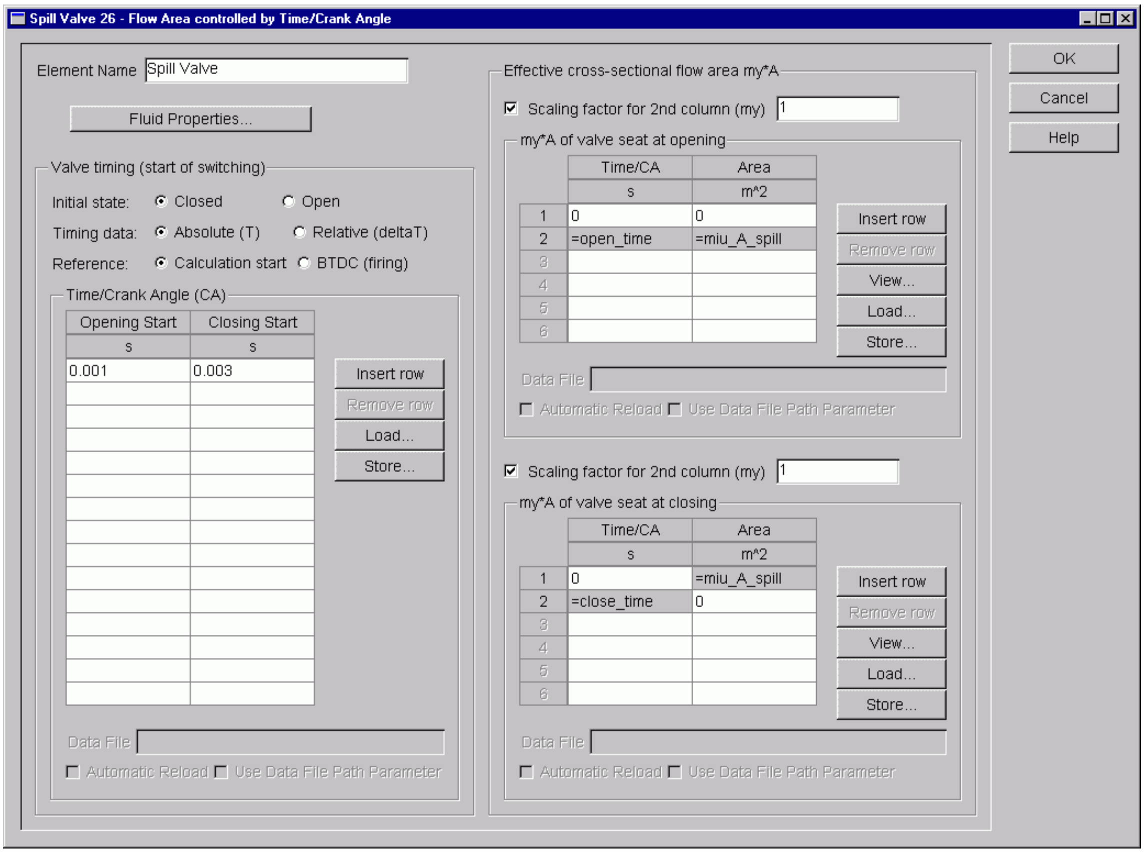


Figure 8-4: Input Dialog of Spill Valve

Create three calculation cases (refer to **Case Explorer** description in *Section 4.8*) and assign the following values to `close_time` and `open_time` parameters:

Case No.	`Close_time (s)`	`open_time (s)`
1	1e-6	3e-6
2	1e-5	2e-5
3	5e-5	5e-5

8.1.1.1. MATLAB® Simulink: General Input Data

In this example, **MATLAB® API: Simulink model** is used. Open input dialog box of **Fill Controller** shown in Figure 8-5. On the left side the dialog element tree is located. In this tree you have three possible choices: **General**, **Input Channels** and **Output Channels**.

Inside **General** input dialog, the complete path (refer to *Section 8.1.2*) and state of an existing Simulink model have to be defined.

Simulink model path can be defined either by typing in the full path or pressing ... button at the right end of the input field and selecting Simulink file (`*.mdl`).

☞

Note: When entering Simulink path, use backslash (\) on PC platforms and slash (/) on Linux/Unix platforms, otherwise BOOST Hydsim will not recognize the path where Simulink model is stored.

Note: It is possible use environmental variables within the Simulink path, e.g. $AVLAST_HOME or $(AVLAST_HOME). If undefined environmental variable is specified, GUI will issue a warning message. If variable name is incorrect, BOOST Hydsim kernel will issue an error message.

State of Simulink model is set to `Continuous` by default. It can be changed in the Combo-box to `Non-continuous` state.

In our example, we specified the path of the Simulink model (refer to *Section 8.1.2*) using the environmental variable $(AVLAST_HOME) and continuous state of this model. If Simulink model is created in **MATLAB®** with **Constant value** element as input and **To Workspace** element as output, then **Input/Output vector** names have to be defined in **General** input dialog.

In the same way, specify Simulink model name and path for **Spill Controller** element.

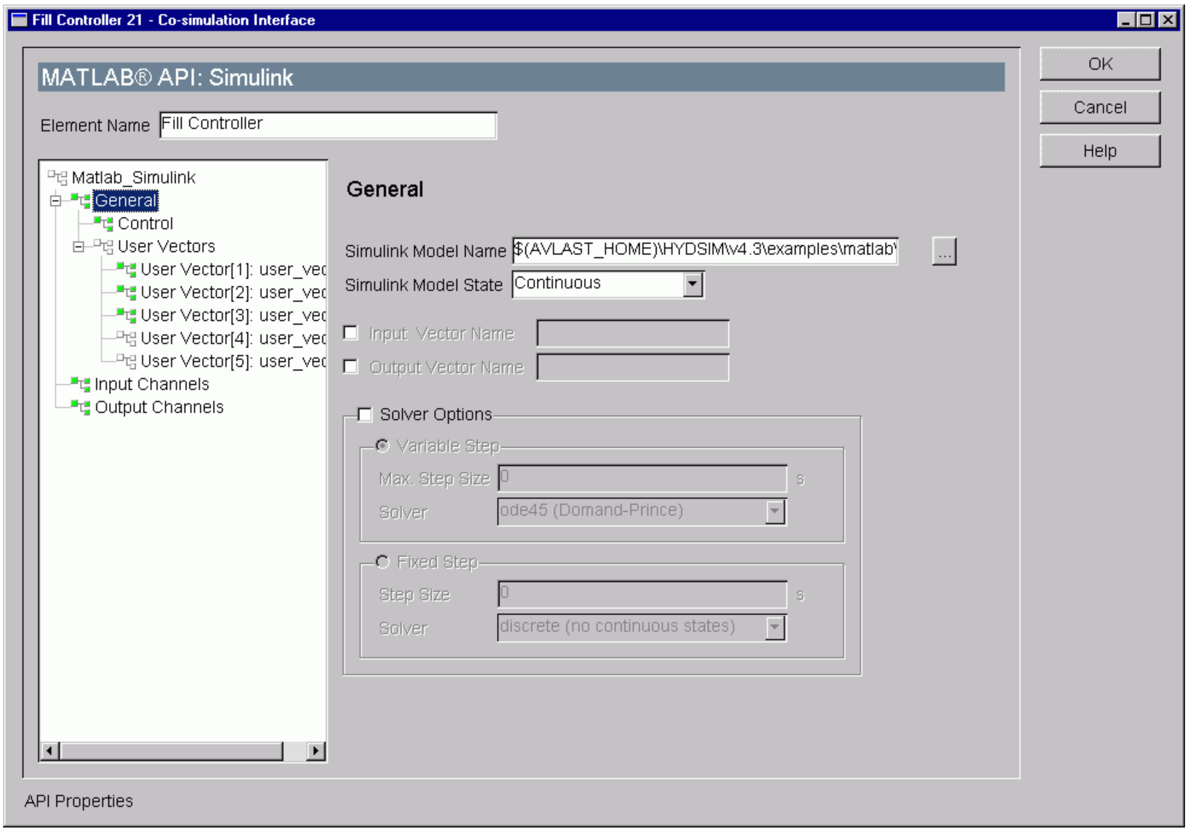


Figure 8-5: Input Dialog of Fill Controller

In **General** input dialog there is a possibility to control the Simulink solver options. For this, activate **Solver Options** button. Default selection is **Variable Step** solvers, 0 for **Max. Step Size** (max. step size will be determined from the start and stop times) and `ode45 (Domand-Prince)` for **Solver type**. For the detailed information refer to *Section 8.1.2.1*.

> **Note:** Depending on the Simulink model state, an appropriate Solver type has to be selected. For example, Simulink model with continuous states is not compatible with discrete solver type. In this case BOOST Hydsim will display an error message generated by MATLAB®.
>
> **Note:** When BOOST Hydsim calculation is completed, SIMULINK™ model will be closed **without saving** it, i.e. all changes from the BOOST Hydsim interface will be neglected.

8.1.1.2. MATLAB® Simulink: Control Input Data

Control subdialog box is used for setting control parameters for BOOST Hydsim - MATLAB® co-simulation. Click the Control button and assign the global parameters `sampling_time` and `response_time` to the respective input data.

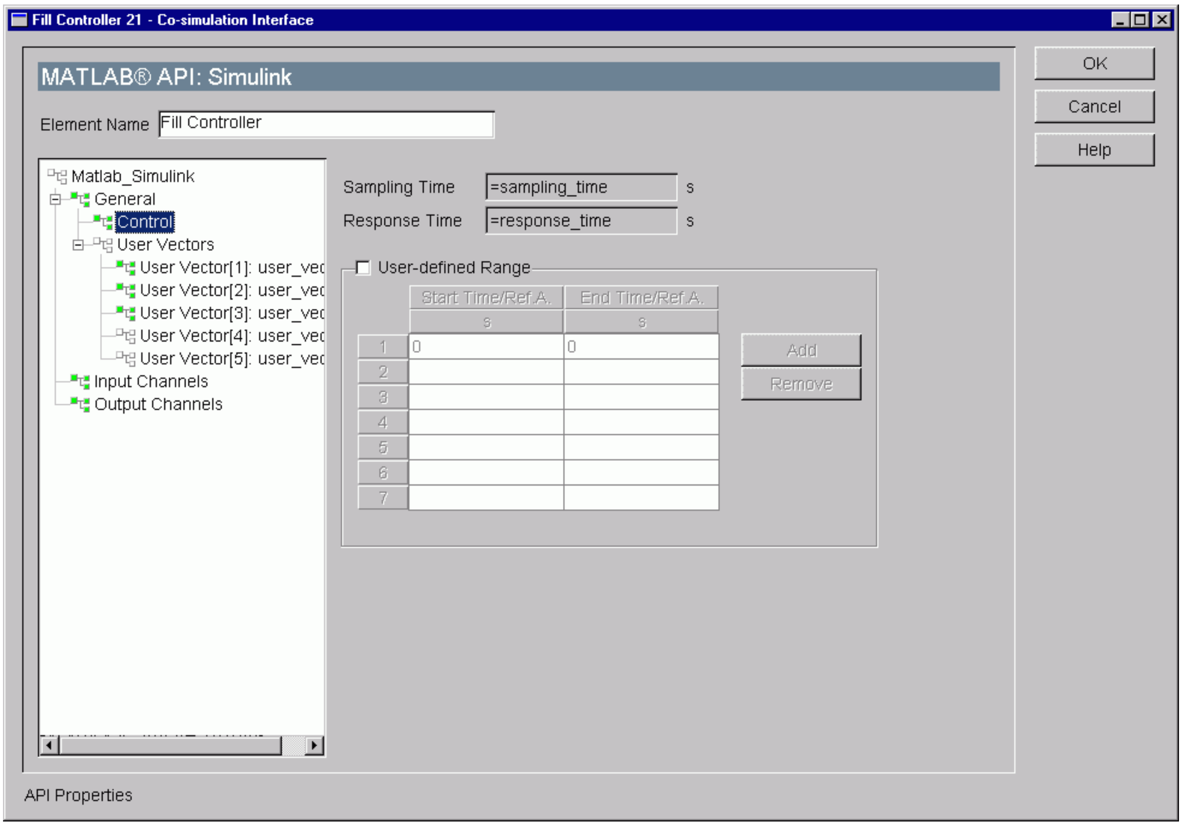


Figure 8-6: MATLAB® API Element Control Dialog

Set these two parameters to the following values in **Case Explorer** dialog:

Case no.	`sampling_time (s)`	`Response_time (s)`
1	5e-4	1e-5
2	0.001	1e-4
3	0.0011	1e-4

Specify the same global parameters for **Spill Controller** element.

Sampling time is typically the scanning time at which MATLAB® Input Channel (Sensor) reads information from the connected BOOST Hydsim element.

Response time is time delay (from sampling event) after which MATLAB® Output Channel (Actuator) sends information to the connected BOOST Hydsim element. Typically it is a reaction time of a physical system represented by the MATLAB® model (due to inertia, wire resistance etc.).

Both parameters **sampling time** and **response time** are set to 0 by default. This implies that data exchange between BOOST Hydsim and MATLAB® will be performed in each calculation step.

Note: If **Sampling/Response time** is smaller than BOOST Hydsim calculation time step, it will be reset to BOOST Hydsim time step. If **Sampling/Response time** is not a multiple of BOOST Hydsim time step, it will be adjusted to the first higher multiple value of time step.

Note: **Response time** must be smaller than or equal to the **Sampling time**. Otherwise BOOST Hydsim will issue an error message.

User-defined Range serves to define independent BOOST Hydsim -MATLAB® will co-simulation intervals. If not active (default), the co-simulation will be performed within the entire BOOST Hydsim calculation interval. Refer to the BOOST Hydsim Users Guide for more information.

8.1.1.3. MATLAB® Simulink: User Vector Input Data

BOOST Hydsim supports passing of user vectors from BOOST Hydsim into MATLAB® Workspace. In this way, user vectors can be used in the Simulink or m-function model. To define a user vector, open e.g. `User vector [1]`. The following dialog box will appear:

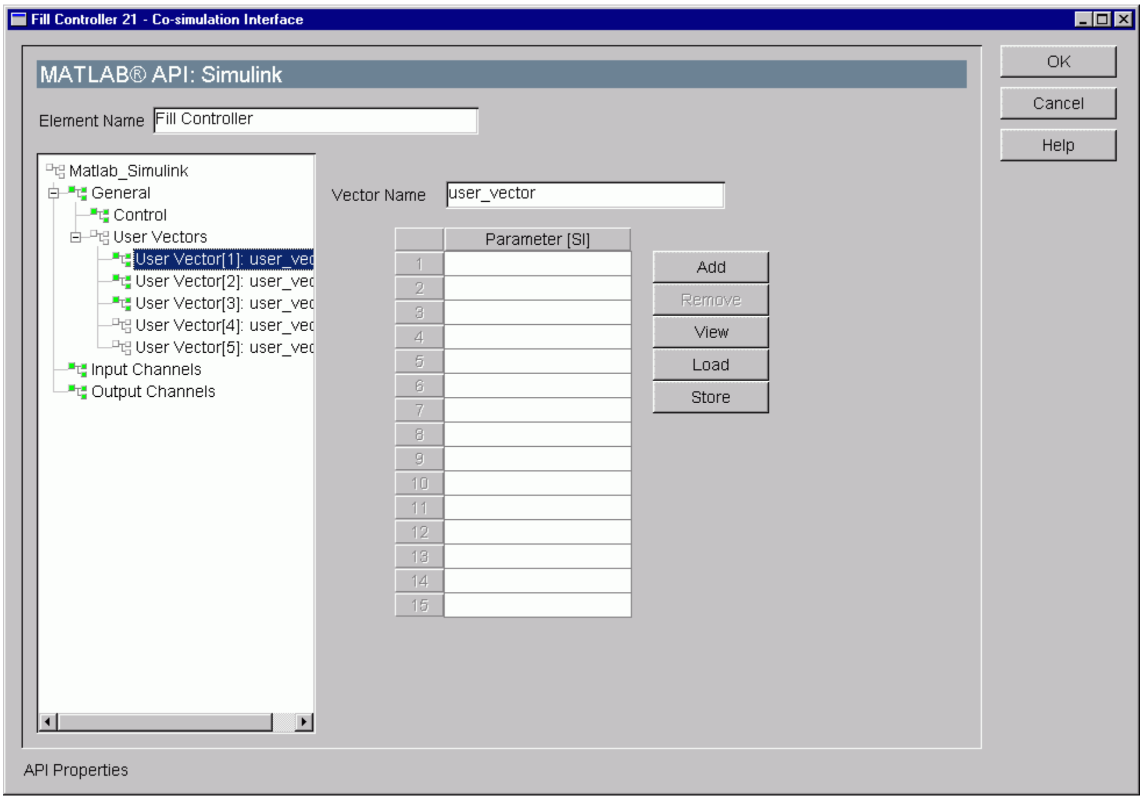


Figure 8-7: Input Dialog of User Vector

Within this dialog box, specify the name in **Vector Name** input field and its entries (parameters) by clicking on Add. Click on View to view all entries.

☞

Note: All entries of user vector must be specified in SI units. Maximum 5 user vectors can be defined. Each vector can have up to 100 entries.

Note: It is not necessary to define user vectors in the successive order because BOOST Hydsim will pass to MATLAB® only the vectors with defined names. If e.g. the 1st, 3rd and 5th user vectors are defined, then only these three vectors (with their names) will be passed to MATLAB® Workspace.

Note: User vector is set into MATLAB® Workspace as one-dimensional matrix. Thus Simulink (or m-function can use its entries for simulation.

Note: Never use same names for user vectors within one or different MATLAB® elements. Otherwise the vector entries will be overwritten.

In our example, three user vectors are defined: `miu_A_fill`, `Valve_time` and `K_f`. First vector possesses only one entry, effective cross-sectional area of **Fill Valve**, which will be used in the Simulink calculation. Second vector contains 7 entries denoting diverse time constants (refer to *Figure 8-14* for Valve timing). These entries are defined as global parameters. Most of them are specified in **Parameters** dialog:

Vector entry no.	Parameter name	Parameter value
1	`Sampling_time1`	`=sampling_time`
2	`Calculation_step1`	`=calculation_step`
3	`Open_time1`	`=open_time`
4	`Close_time1`	`=close_time`
7	`Response_time1`	`=response_time`

Double definition of same parameters (with two different names) is necessary due to unit system: user vector requires a SI unit, so a parameter with an arbitrary unit cannot be used there. All parameters are already defined, except `calculation_step` assigned in **Calculation Control** dialog box and set to `3e-7 s`.

Remaining two parameters are defined in the **Case Explorer** as follows:

Vector entry no.	Parameter name	Parameter values (s)		
		Case 1	Case 2	Case 3
5	`open_delay`	1e-5	4e-5	5e-5
6	`close_delay`	1e-5	4e-5	5e-5

The third user vector contains PID controller parameters K_P, K_I and K_D (refer to *Section 8.1.2* for calculation of their values).

For **Spill controller** the following user vectors are used:

User vector name	Vector entry no.	Parameter name	Parameter value
`miu_A_Spill`	1	miu_A_spill1	=miu_A_spill

User vector name	Vector entry no.	Vector entry value
`K_s`	1	78.4
	2	2.4e+9
	3	6.533e-7

8.1.1.4. MATLAB® Simulink: Input Channels Data

Select **Input Channels** in the dialog element tree to open the following:

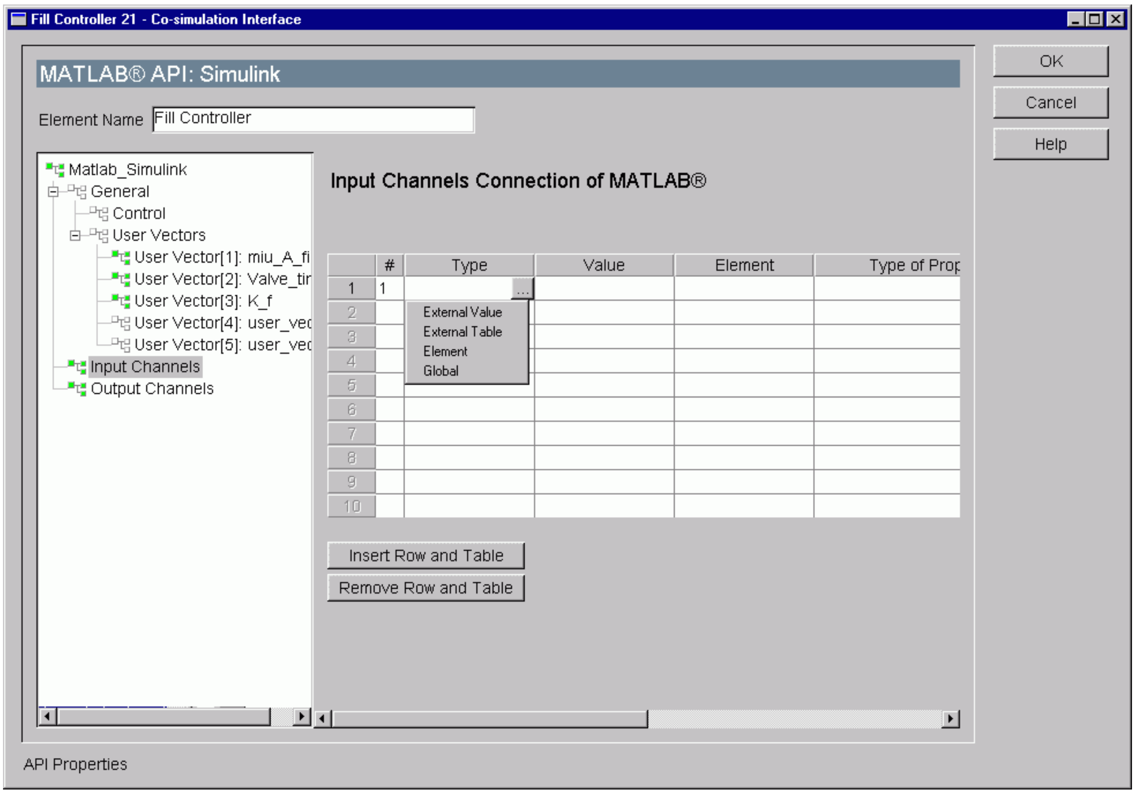


Figure 8-8: Input Dialog of Input Channels

Select [Insert Row and Table] to enter the input parameters. To select the appropriate parameter for the input channel, click inside the input field and select a type of input channel from the submenu.

For the first channel select **External Value**, then enter a constant value in the **Value** column. Set it to 1500e5 Pa in SI unit (1500 bar) which is our reference pressure in Rail Volume.

☞ **Note:** **External Value** and all values in **External Table** must be specified in SI units.

For the second channel select **Element**, then open submenu in the Element column and select **Rail Volume** element. Click into **Type of Properties** column and select there **El. output parameter**. Then click into Property column and select **pressure in volume [Pa]**. In this way, BOOST Hydsim will send pressure (in Pa) of the Rail Volume in the second input channel to the Simulink model.

☞ **Note:** In **Element** column, BOOST Hydsim lists only those elements which are connected to the corresponding MATLAB® element by wire connection.

In the same manner select other input channels as follows:

Channel #	Type	Value	Element	Type of Property	Property
3	Element		Rail Volume (20)	El. local fluid properties	bulk modulus [N/m^2]
4	Element		Rail Volume (20)	El. output parameter	actual volume [m^3]
5	Element		Fill Valve (25)	El. local fluid properties	fluid density [kg/m^3]
6	Element		Pump Volume (23)	El. output parameter	pressure in volume [m^3]
7	Global				time [s]

In the **#** column the numbers are the Port numbers for the Simulink model (refer to *Section 8.1.2*).

☞ **Note:** Changing numbers in **#** column would be necessary if e.g. the user modifies the numbering of input and/or output channels in Simulink model.

For **Spill Controller** set the following input channels:

Channel #	Type	Value	Element	Type of Property	Property
1	External value	1500e5			
2	Element		Rail Volume (20)	El. output parameter	pressure in volume [Pa]
3	Element		Rail Volume (20)	El. local fluid properties	bulk modulus [N/m^2]

4	Element		Rail Volume (20)	El. output parameter	actual volume [m^3]
5	Element		Spill Valve (26)	El. local fluid properties	fluid density [kg/m^3]
6	Element		Spill Pressure (13)	El. output parameter	pressure [Pa]
7	Global				time [s]

8.1.1.5. MATLAB® Simulink: Output Channels Data

Select **Output Channels** in the dialog element tree and specify output channels in the same way as input channels. Select the following properties:

1st channel – Fill Valve (25) as Element, El. input parameters as Type of Property and switching time T1/delta T1 [s] as Property

2nd channel – Fill Valve (25) as Element, El. input parameters as Type of Property and switching time T2/delta T2 [s] as Property

> ☞ **Note:** Number of selected input/output channels has to be the same as the number of input/output channels in the Simulink model. Otherwise, BOOST Hydsim will issue an error message. However, if input/output vector names are used, BOOST Hydsim will not control number of selected input and output channels.

For **Spill Controller** set the following output channels:

1st channel – Spill Valve (26) as Element, El. input parameters as Type of Property and switching time T1/delta T1 [s] as Property

2nd channel – Spill Valve (26) as Element, El. input parameters as Type of Property and switching time T2/delta T2 [s] as Property

8.1.2. Creating Simulink model

Creating of a Simulink model requires the SIMULINK™ installation of MATLAB® software package. Here we assume that the user is familiar with the SIMULINK™ application.

In our example we use PID controllers for regulating the pressure in the Rail Volume around constant pressure of 1500 bar. As input variable, it uses pressure difference between actual pressure in Rail Volume and rated pressure. Based on this difference, the algorithm calculates the opening time of Fill Valve. Opening time is calculated from the continuity equation *(8.1.1)* and Bernoulli equation *(*8.1.2*)*. Opening time has to be large enough to cover the minimum opening and closing time delays and ramps (refer to *Figure 8-14*).

$$\Delta p = \frac{E}{V} \cdot \dot{Q} \,, \qquad (8.1.1)$$

where:

Δp .... pressure difference

E ....bulk modulus of the fluid

V ....actual volume of Rail Volume

$\dot{Q}$ .... cumulative volumetric flow rate

$$\dot{Q} = \mu A \cdot \sqrt{\frac{2 \cdot \Delta p}{\rho}} \cdot \Delta T \,, \qquad (8.1.2)$$

where:

μA .... effective cross-sectional area of Fill Valve

ρ .... fluid density

ΔT .... opening time

To determine all three PID controller elements (Proportional, Integral and Derivative), `Ziegler Nichols` tuning method (`open loop reaction rate`[9]) is used. This method, known as the "reaction curve" method, is illustrated in Figure 8-9.

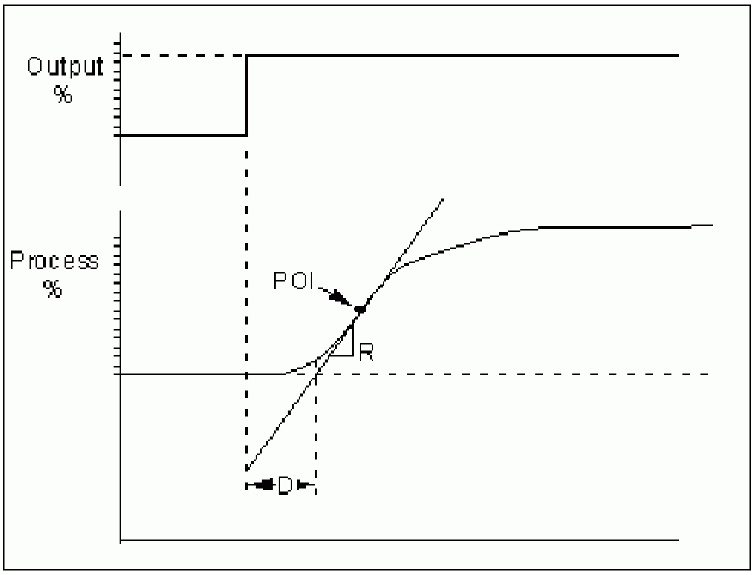


Figure 8-9: Controlled Process Graph

Here the following notations are used:

X (%) .... change of output (linear relationship between output and process is assumed)

R (%/min) ....rate of change at the point of inflection (POI)

[9] John Shaw, The PID Control Algorithm: How it Works and how to Tune It

D (min) ....time till interception of tangent line and original process value

In our example, we use the following values:

$$X = \frac{\Delta p}{p_{set}} \cdot 100(\%) = 0{,}6667\% ,$$

where:

$\Delta p(assumed)$ ..... pressure difference (10 bar),

p_{set} .... rated pressure (1500 bar).

$$R = \frac{1}{\Delta T} = 612245 \frac{\%}{\min} ,$$

where:

ΔT .... reaction time (Interaction Time-2*D=1.6333e-6 min)

$D = 1{,}667 \cdot 10^{-8} \min(assumed)$

The Proportional, Integral and Derivative constants are calculated by:

Proportional: $K_P = 1{,}2 \cdot \frac{X}{D \cdot R} = 78.4$.

Integral: $K_I = Gain \cdot \mathrm{Re}\,set = 2.4 \cdot 10^9$,

where:

$Gain = K_p$

$\mathrm{Re}\,set = \frac{0.5 \cdot}{D}$

Derivative: $K_D = Gain \cdot Derivative = 6.533 \cdot 10^{-7}$,

where

$Derivative = 0.5 \cdot D$.

Figure 8-10 shows the Simulink model of **Fill Controller** based on the above definitions.

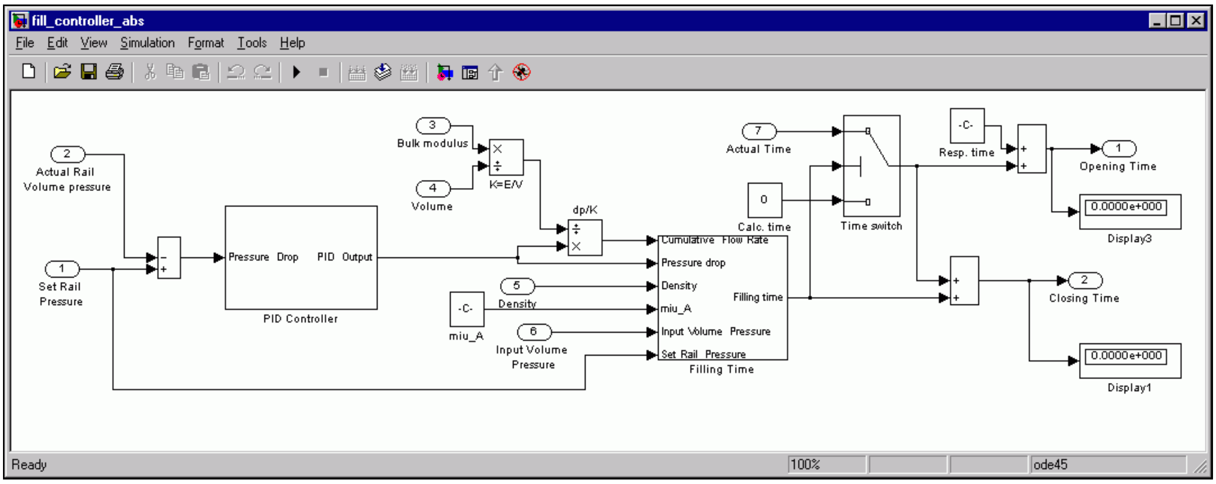


Figure 8-10: Simulink Model of Fill Controller

Within this Simulink model (file `fill_controller_abs.mdl`), three user vectors (refer to *Section 8.1.1.3*) are used for PID coefficients. They are defined in BOOST Hydsim model as vectors `K_f`, effective cross-sectional Area `miu_A` and `Valve_time`.

PID controller itself is defined by the subsystem `PID Controller` (refer to *Figure 8-11*) with `K_f(1)` as proportional coefficient, `K_f(2)` as integral coefficient and `K_f(3)` as derivative coefficient.

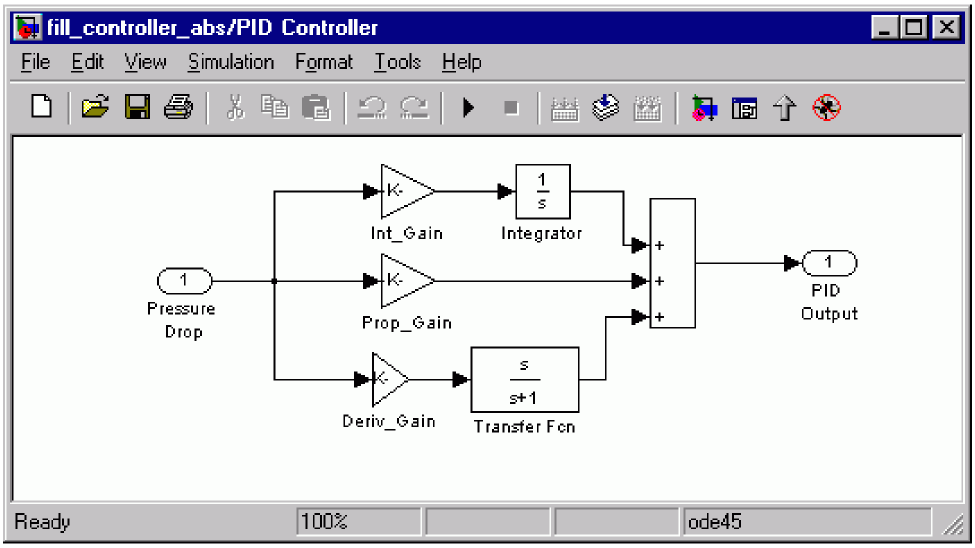


Figure 8-11: Simulink Model of PID Controller

In the same way, the Simulink model of **Spill Controller** shown in *Figure 8-12* is built up. It contains analogous PID components, but with the opposite calculation algorithm for pressure difference

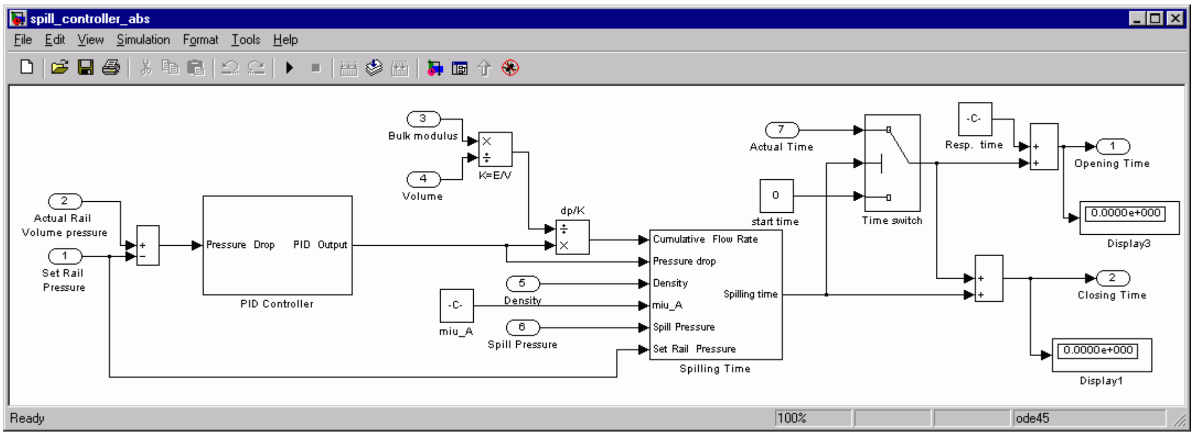


Figure 8-12: Simulink Model of Spill Controller

Calculation of opening time of **Fill Valve** is performed by `Filling Time` subsystem shown in *Figure 8-13*. This subsystem calculates respective opening time `Tcalc` (refer to *Figure 8-14*) according to the Bernoulli equation *(8.1.2)* and predefined timing characteristics of the valve movement. All timing data are defined in the user vector `Valve_time`.

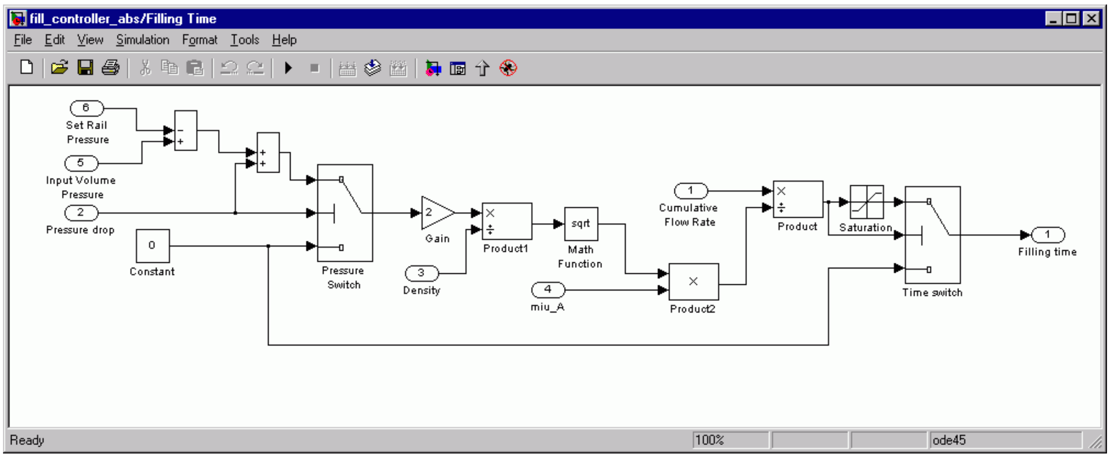


Figure 8-13: Simulink Model for Filling Time Calculation

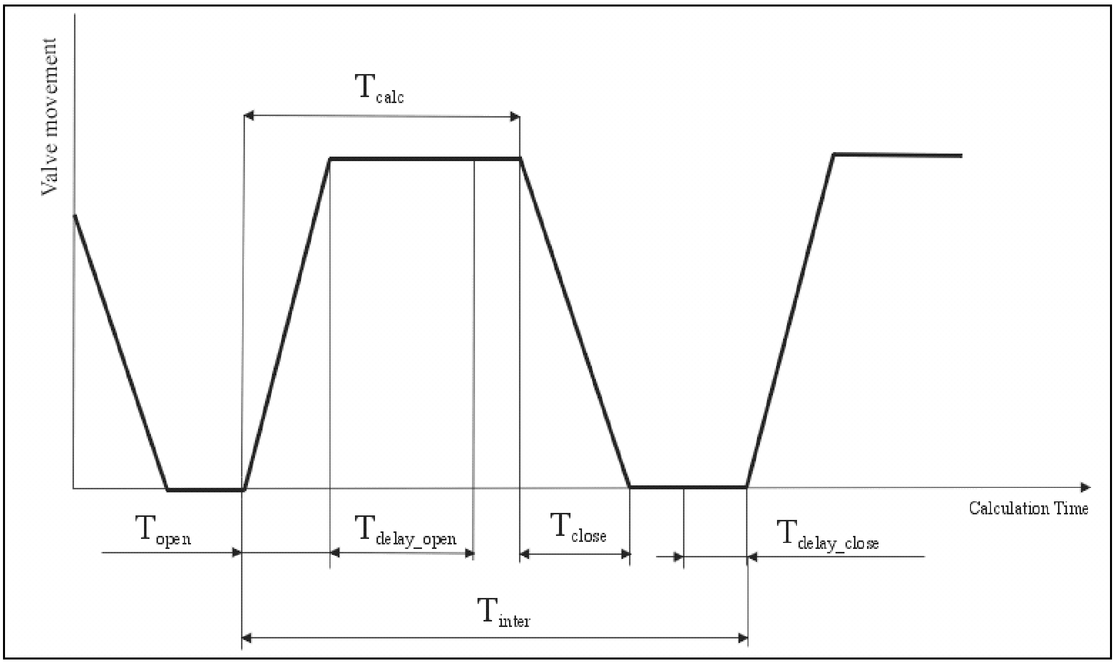


Figure 8-14: Timing of Valve Movement

Calculated opening (filling) time has to satisfy the following bounds:

$$(T_{calc})_{\min} = T_{open} + T_{delay\_open},$$

$$(T_{calc})_{\max} = T_{inter} - (T_{close} + T_{delay\_close}),$$

where:

T_{calc} .... calculated filling time

$(T_{calc})_{\min}$ .... minimum possible filling time

$(T_{calc})_{\max}$ .... maximum possible filling time

T_{open} .... opening time (from 0 to max. lift)

T_{delay_open} .... min. opening delay (specified)

T_{inter} .... sampling (interception) time

T_{close} .... closing time (from max. lift to 0)

T_{delay_close} .... min. closing delay (specified)

8.1.2.1. Solver options

After having created the Simulink model, set the calculation parameters: Solver and Workspace I/O options. Open it from **Simulation | Simulation parameters** PullDown menu in the Simulink model of **Fill Controller**.

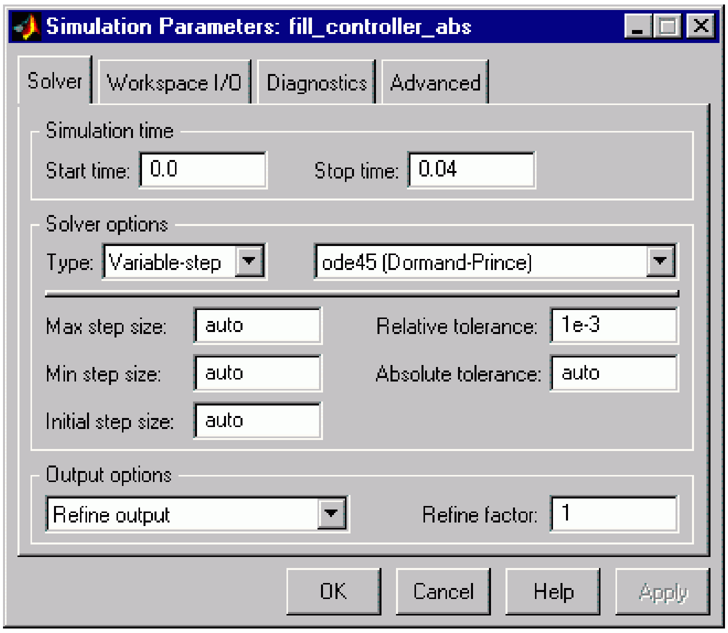


Figure 8-15: Solver Options

In the **Solver** window shown in Figure 8-15, **Simulation time** (Start and Stop) has no meaning because BOOST Hydsim will reset it during calculation. The entries for **Solver Options** can be either set there to the desired values or changed in BOOST Hydsim during pre-calculation (refer to *Section 8.1.1.1*).

User can choose incrementation between `Variable-step` (modified step size during simulation) and `Fixed-step` solver (constant step size during simulation) and select type of `Solver`. The default option is `ode45` for the variable-step Solvers and `discrete` for the fixed-step Solvers.

For variable-step solvers, the user can set the maximum and initial step size parameters. By default, indicated by the value `auto`, these parameters are determined automatically. The **Max. step size** parameter controls the largest time step the solver can take. The default step size is calculated from the Start and Stop times as:

$$\texttt{Max. step size} = \frac{t_{stop} - t_{start}}{50}.$$

The **Initial step size** parameter is a suggested first step size.

For fixed-step solvers, the user can set the **Fixed step size** which is set to `auto` by default (default value is determined according to the same equation as default **Max. step size** for variable-step solvers).

BOOST Hydsim will set **Output options** to `Refine output` with **Refine factor** `1` during co-simulation because it gets only the last values from the output vectors as simulation results.

8.1.2.2. Workspace options

Select the **Workspace I/O** tab in **Simulation parameters** dialog box.

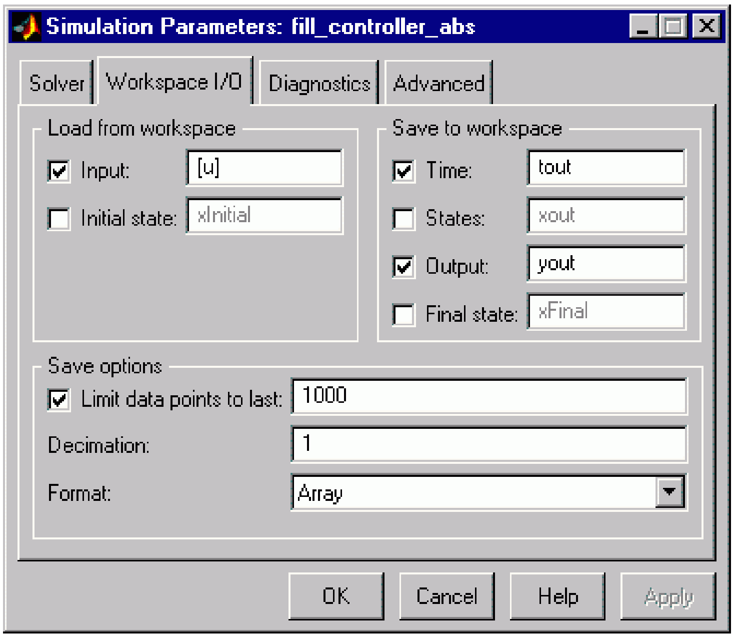


Figure 8-16: Workspace I/O Options

It is not necessary to select any parameter in this dialog box because BOOST Hydsim will set all required parameters. If the Simulink model is created with input port elements (no **Input vector** name - refer to *Figure 8-5*), it will set **Load from workspace** as **Input vector** `[u]`, otherwise this parameter will be disabled. External Input vector `u`, will have one row with `n+1` columns, where `n` is the total number of the model input ports (first entry is the simulation start time).

If `Continuous` model state is selected (refer to *Figure 8-5*), **Final state** and **Initial state** will be set in MATLAB® Workspace as `FinalState_`***m*** during co-simulation, where ***m*** is the order number of MATLAB® element in BOOST Hydsim model. It means that BOOST Hydsim will take `Final State` from previous simulation as `Initial State` for the actual co-simulation (as `Initial State` at co-simulation start is 0).

Time, States and Output vectors are set to `[time]`, `[state]` and `[out]` vectors, respectively.

8.1.3. Output Parameters

BOOST Hydsim supports the viewing of all input and output channel parameters of MATLAB® API element in Impress Chart post-processor. To perform this, select desired output parameters from the **Element | Store Results** menu. These output results may be used to control the BOOST Hydsim -MATLAB® co-simulation.

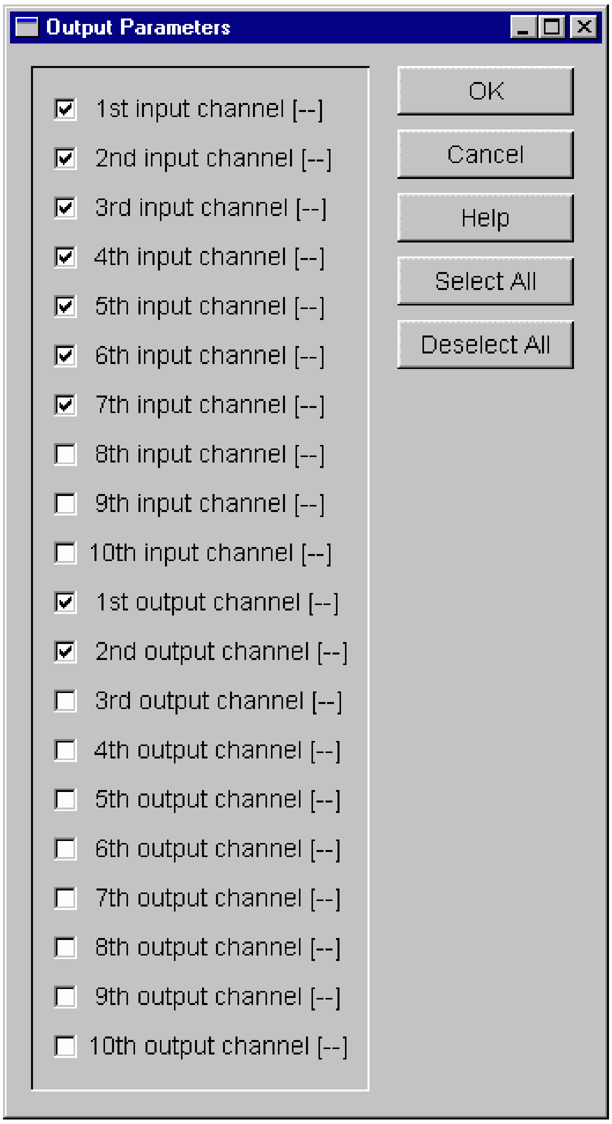


Figure 8-17: Output Data of MATLAB API Element

8.1.4. Running the Calculation

There are no **Initial Conditions** in MATLAB® API: Simulink element.

For performing the calculation, refer to *Section 4.4*.

In this example, Opening/Closing times of Fill and Spill Valves are defined in absolute time domain. Alternatively, it is possible to specify them in the relative time domain. For this, switch **Timing data** button in the input dialog of Fill and Spill Valve to **Relative (delta T)** as shown in Figure 8-18. Adjust the Simulink model accordingly (refer to *Figure 8-19*).

Save the adjusted Simulink model and BOOST Hydsim model under different names, e.g. `fill_controller_rel.mdl` and `comrail2wv_rel_mdl_1.hyd` respectively.

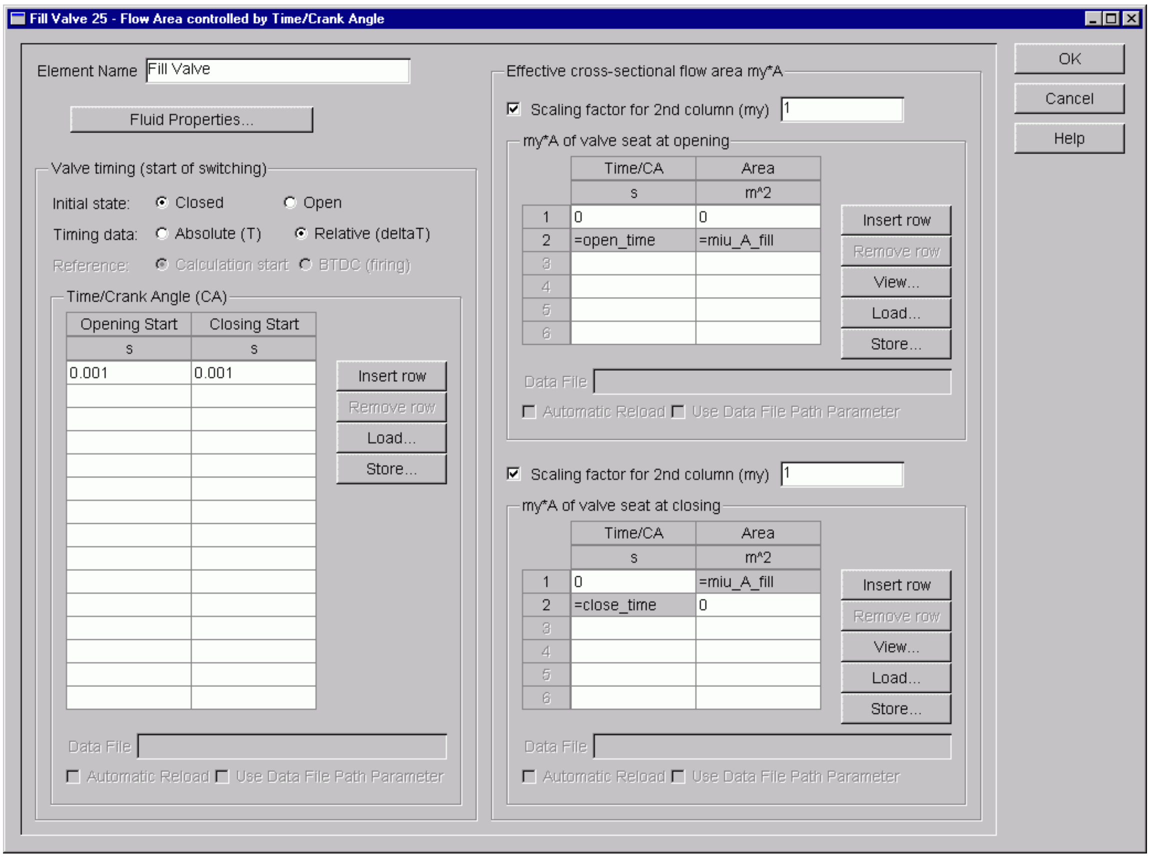


Figure 8-18: Input Dialog Box of Fill Valve with Relative Timing Data

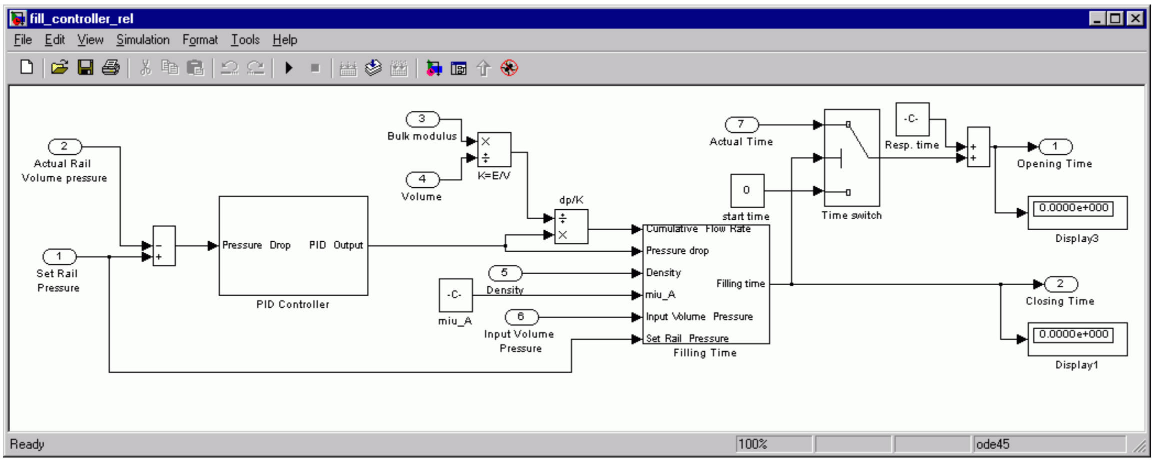


Figure 8-19: Simulink Model of Fill Valve with Relative Timing

8.1.5. Calculation Results

In this section, the calculation results of three cases defined in **Case Explorer** are shown, for the BOOST Hydsim model with absolute timing values. Remember, each case contains individual sampling and response times and different timing constants of the Fill and Spill Valves.

To view the calculation results, select **Show Results** from **Simulation** Pulldown menu. The new Impress Chart window with the **Result** tree of the active BOOST Hydsim model will pop up.

Figure 8-20 shows the pressures in `Rail Volume`, `Pump Volume` and `Branch Volume`. Figure 8-21 depicts effective flow area of `Fill Valve`, for each case separately. Analysis of the calculation results is left here for the user as an exercise.

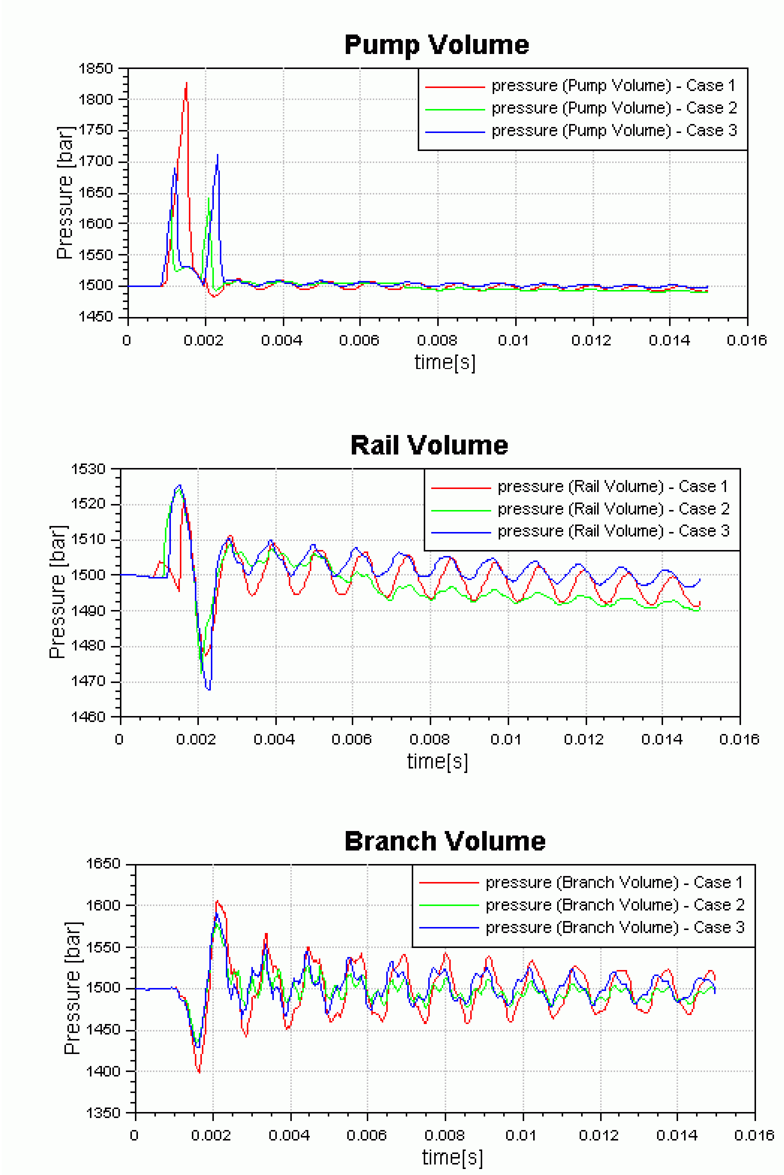


Figure 8-20: Pressure in Pump, Rail and Branch Volumes

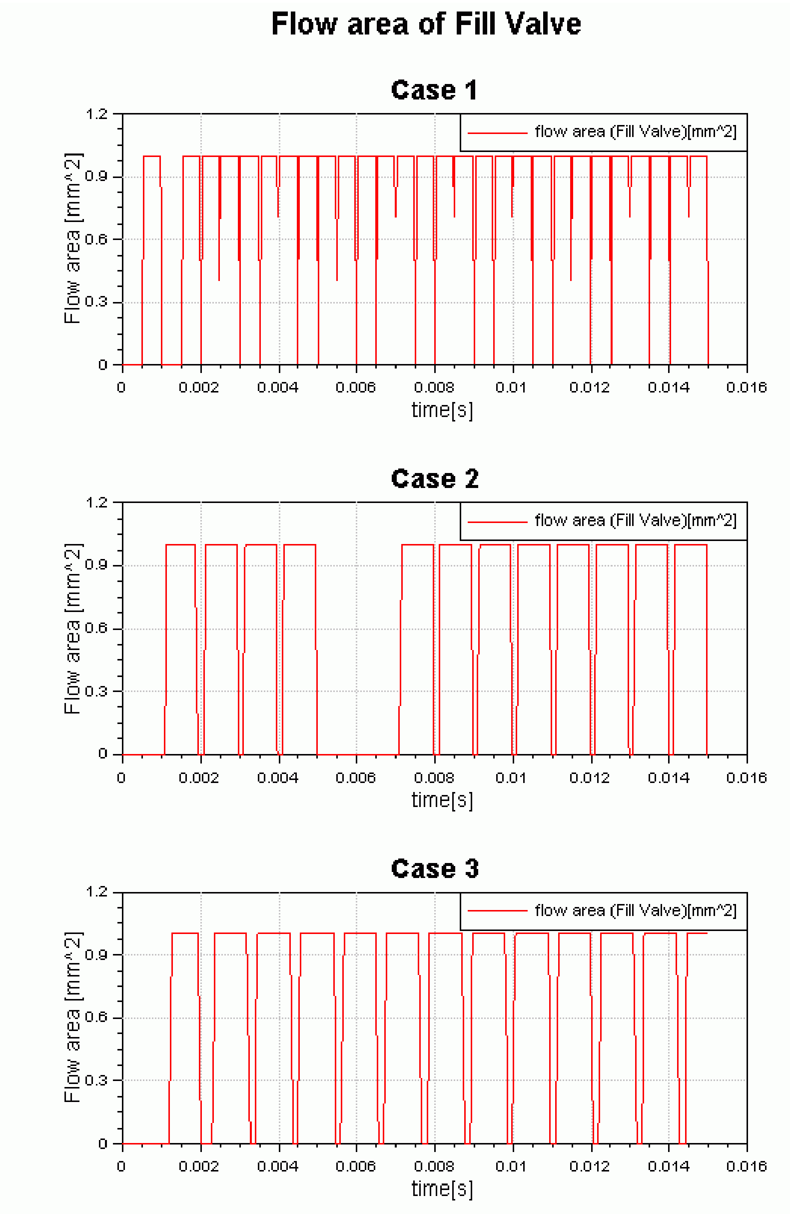


Figure 8-21: Flow Area of Controlled Fill Valve

8.1.6. Running the Calculation with Case Sets

We use the basic model `comrail2wv_abs_mdl_1.hyd` for extending it with three **Case Sets**. Assume we need to run the model with three different rail pressures, and within each pressure case we have to vary the volume of fuel rail. An intelligent way to set up and perform such a task is to use **Case Sets**.

Rail pressure is used in the Pump Volume (element No. 23), Rail Volume (No. 20), Branch Volume (No. 2), Control Volume (No. 6) and Valve Volume (No. 7). Define initial pressure in these volumes to global parameter `Rail_pressure`.

Assign global parameter `Pump_volume` to the Initial volume of Pump Volume, `Rail_volume` to the Initial volume of Rail Volume. Next, assign local parameter `Over_pressure` to the Opening/closing pressure of Over-pressure Valve (No. 24).

Open input dialog boxes of Fill and Spill Controller and assign External Value in the first Input Channel/Value column to global parameter Rail_`pressure1`.

Create three **Case Sets** and set the parameter `Rail_pressure` to 1350, 1600 and 1800 bar, respectively. Rename the **Case Sets** to `Rail_pressure_135MPa`, `Rail_pressure_160MPa` and `Rail_pressure_180MPa`, respectively. Under each **Case Set** create three **Cases** with different values for `Rail_volume`. Supply appropriate name for each case as shown in table below.

Case Set	Case name	Rail_volume (mm3)	Rail_pressure (Pa)
Rail_pressure_135MPa	Rail_volume_10cm3	10000	135e6
	Rail_volume_15cm3	15000	
	Rail_volume_20cm3	20000	
Rail_pressure_160MPa	Rail_volume_15cm3	15000	160e6
	Rail_volume_20cm3	20000	
	Rail_volume_25cm3	25000	
Rail_pressure_180MPa	Rail_volume_20cm3	20000	180e6
	Rail_volume_25cm3	25000	
	Rail_volume_30cm3	30000	

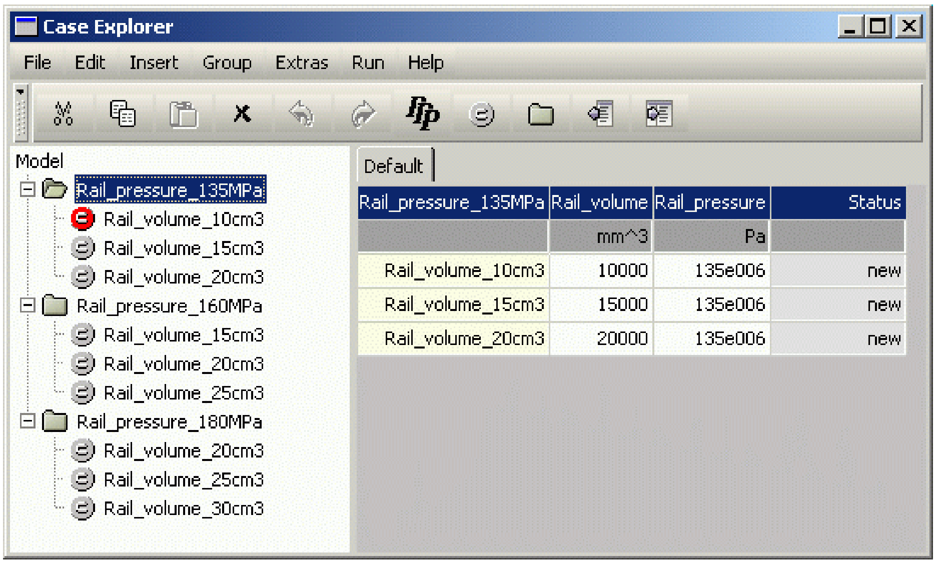


Figure 8-22: Three Case Sets

Within **Model Parameters** dialog set global parameter `Pump_volume` to `Rail_volume/10`, global parameter `Rail_Pressure1` to `Rail_pressure` and local parameter `Over-pressure` to 180 MPa.

Set parameters `sampling_time` to 0.001 s and `response_time` to 1e-4 s.

Save the model under different name, e.g. `comrail2wv_abs_3case_sets.hyd`. Run the calculation for all three **Case Sets**, i.e. nine cases altogether.

Figure 8-23 shows the results for the Rail Volume pressure. Each layer represents one **Case Set** and contains three curves corresponding to different Rail Volume sizes.

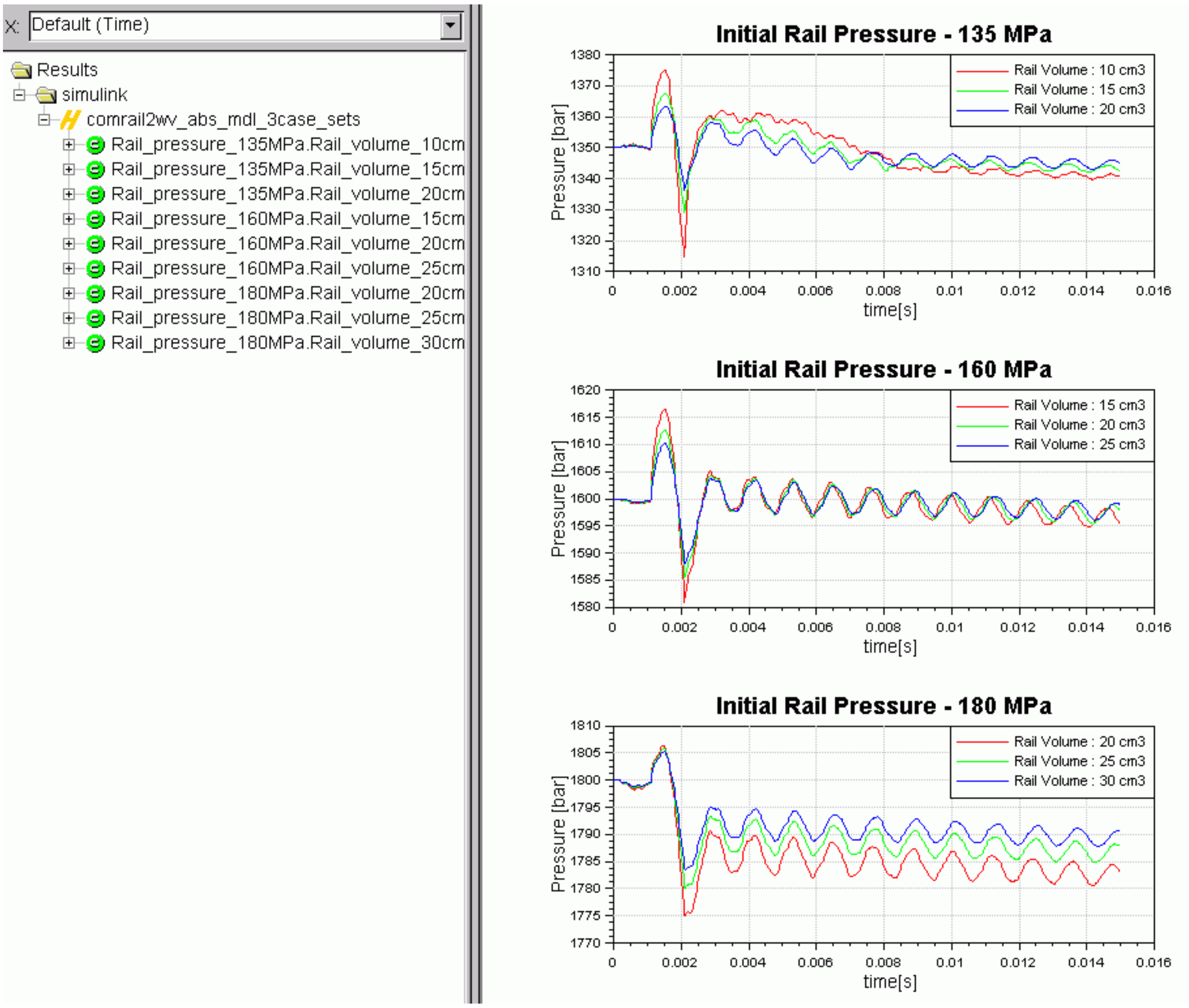


Figure 8-23: Pressure in Rail Volume for Three Case Sets

8.2. Example with MATLAB® m-function

In this example, an application of **MATLAB® m-function** element is shown. It represents a P-controller in BOOST Hydsim model of a simple pipe-volume-orifice system.

The model of the system is shown in *Figure 8-24*. It consists of Input Pressure (boundary condition) with variable pressure, Line, Controlled Volume, Control Throttle and Leak Pressure (1 bar). P-Controller (**m-function** element) has to regulate pressure in the Controlled Volume so that it does not exceed the limiting value (1250 bar in our case). Control parameter is the diameter of the Control Throttle.

If the pressure in the system increases beyond the limiting value, P-Controller has to open Control Throttle, thus initiating the fluid flow from Controlled Volume to Leak Pressure. As soon as the pressure drops below the limiting value, P-Controller has to close the Control Throttle again.

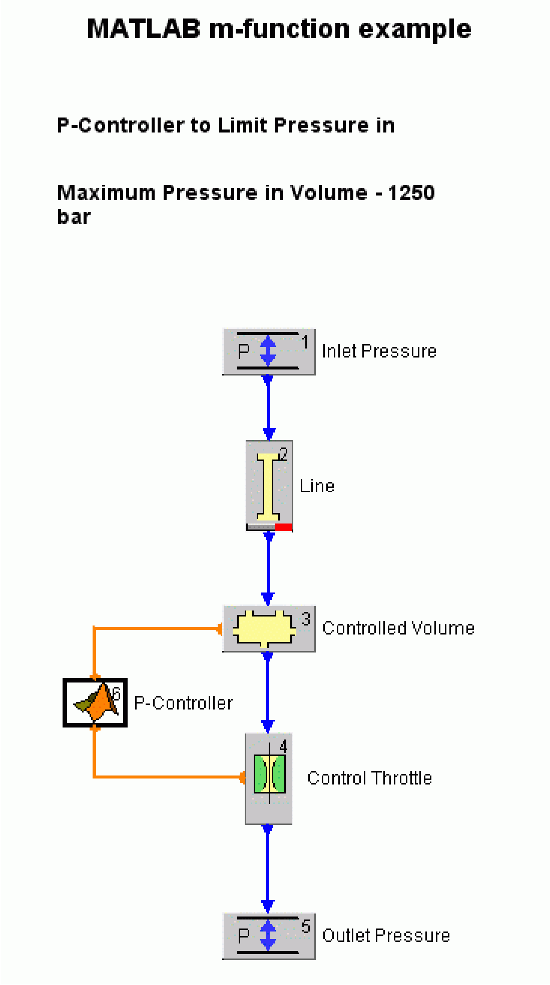


Figure 8-24: BOOST Hydsim Model with P-Controller as MATLAB® m-function

8.2.1. Creating m-function

The m-function is created with the M-file Editor of MATLAB® software. Here we assume that the user is familiar with the M-file Editor.

As stated earlier, in our example we use P-controller to regulate pressure in Controlled Volume. As Input variable, the controller uses pressure difference between actual pressure of Controlled Volume and rated (limiting) pressure. According to this difference the cross-section diameter of Control Throttle is calculated.

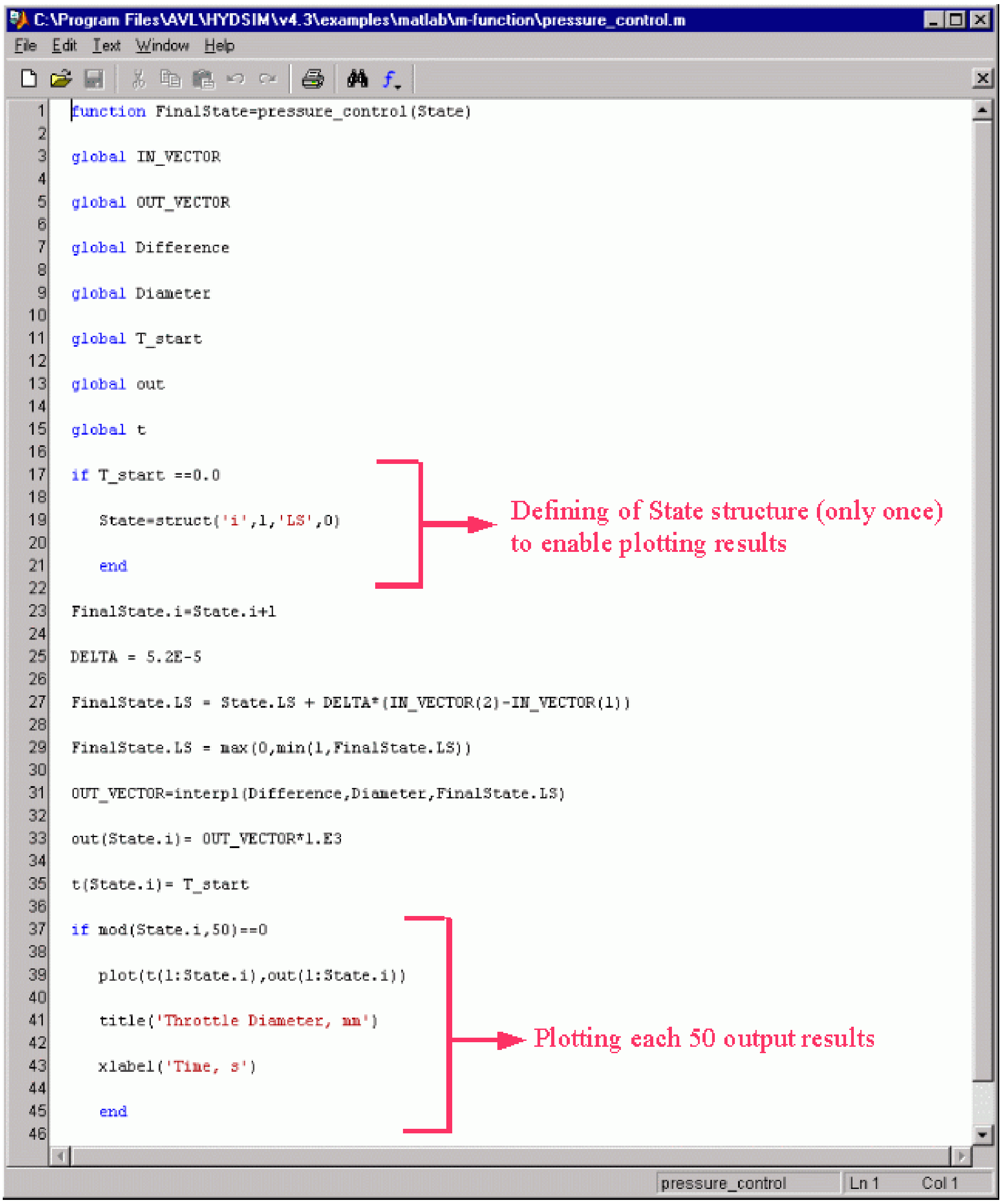


Figure 8-25: M-function for P-Controller

The screen shot of the m-function in shown in Figure 8-25. The m-function has to be declared as `function FinalState=function name(State)` because BOOST Hydsim uses this statement while calling m-function. For each calculation time step, program will assign `FinalState` variable to `State` variable, which will be used as initial state for the next calculation step.

Input and output vectors must be declared as global variables. These vectors must be defined in BOOST Hydsim with the same names (refer to *Section 8.2.2.1*).

In this example, two additional vectors `Difference` and `Diameter` are used. These vectors define the functional relationship: effective diameter of Controlled Throttle vs. pressure difference.

According to it and calculated `FinalState` variable, m-function interpolates the value for output vector (cross-section diameter of Control Throttle). The above vectors are defined in BOOST Hydsim model as user vectors (refer to *Section 8.2.2.1*).

The last part of the m-function serves to plot the calculated diameter of Control Throttle vs. time. For visualization purpose each 50th step is plotted during co-simulation (refer to *Figure 8-30*).

Save the m-function on file under the name `pressure_control.m`.

8.2.2. Input Data

In this section, the input data of Control Throttle and P-Controller will be discussed.

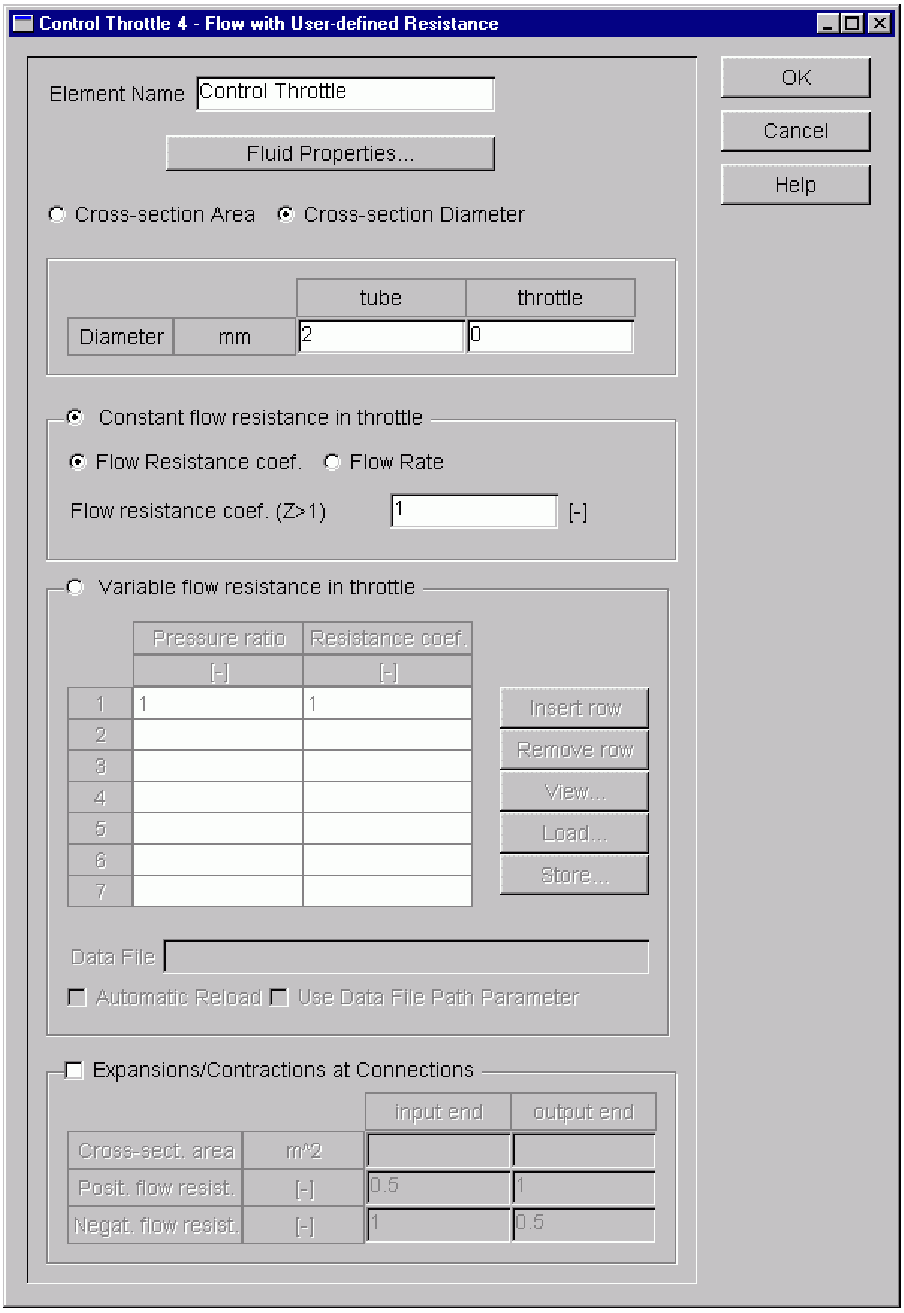


Figure 8-26: Input Data of Control Throttle

Input dialog box of Control Throttle is shown in Figure 8-26. At calculation start Control Throttle is closed (its cross-section diameter is set to 0). For simplicity, the flow resistance coefficient is set to 1, i.e. the calculated throttle diameter is not a geometric but an effective diameter. If the pressure in Controlled Volume increases beyond 1250 bar, P-Controller will calculate the new effective diameter of Control Throttle.

8.2.2.1. MATLAB® m-function: Input Data

Open the input dialog box of P-controller. In General subdialog shown in Figure 8-27, define the complete path name of m-function and the names of input/output vectors. These vectors have to correspond to the respective vectors in the m-function (refer to *Section 8.2.1*).

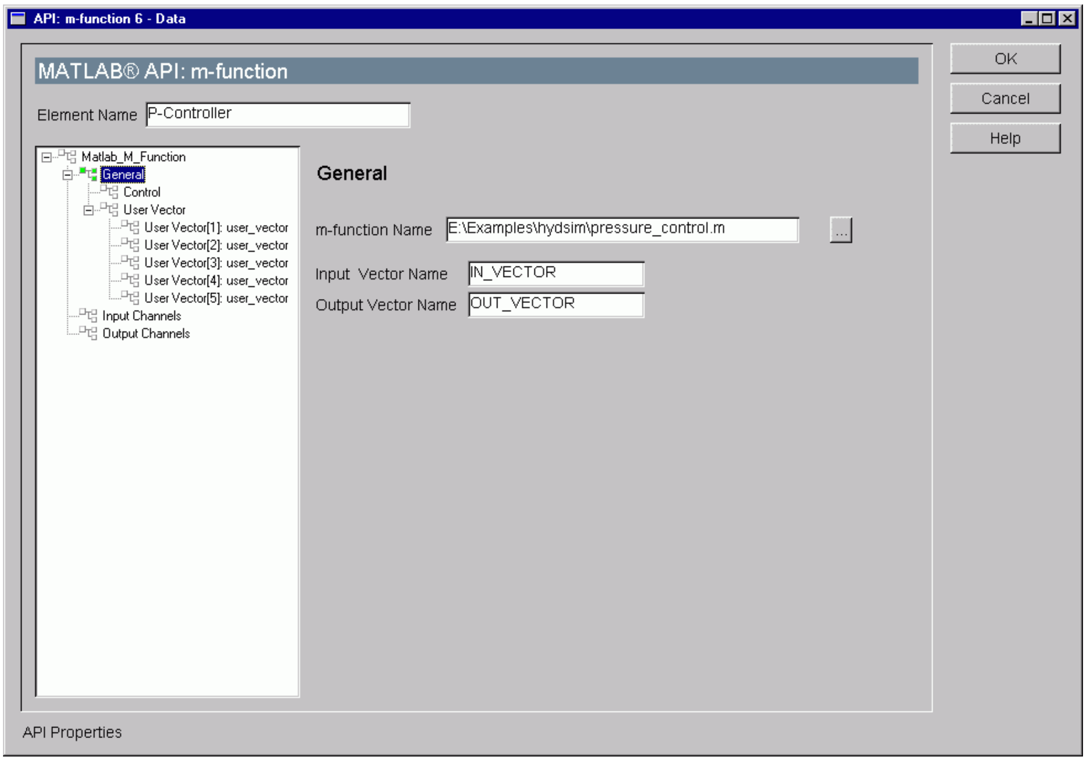


Figure 8-27: General Input Dialog Box of P-Controller

In Control subdialog (Figure 8-28), assign global parameters `sampling_step` and `response_step` to the corresponding input data (refer to *Section 8.1.1.2* for their definition). Create three cases and set the above parameters to the following values:

Case no.	`response_step` (s)	`sampling_step` (s)
1	2e-7	5e-7
2	4e-7	1e-6
3	5e-7	2e-6

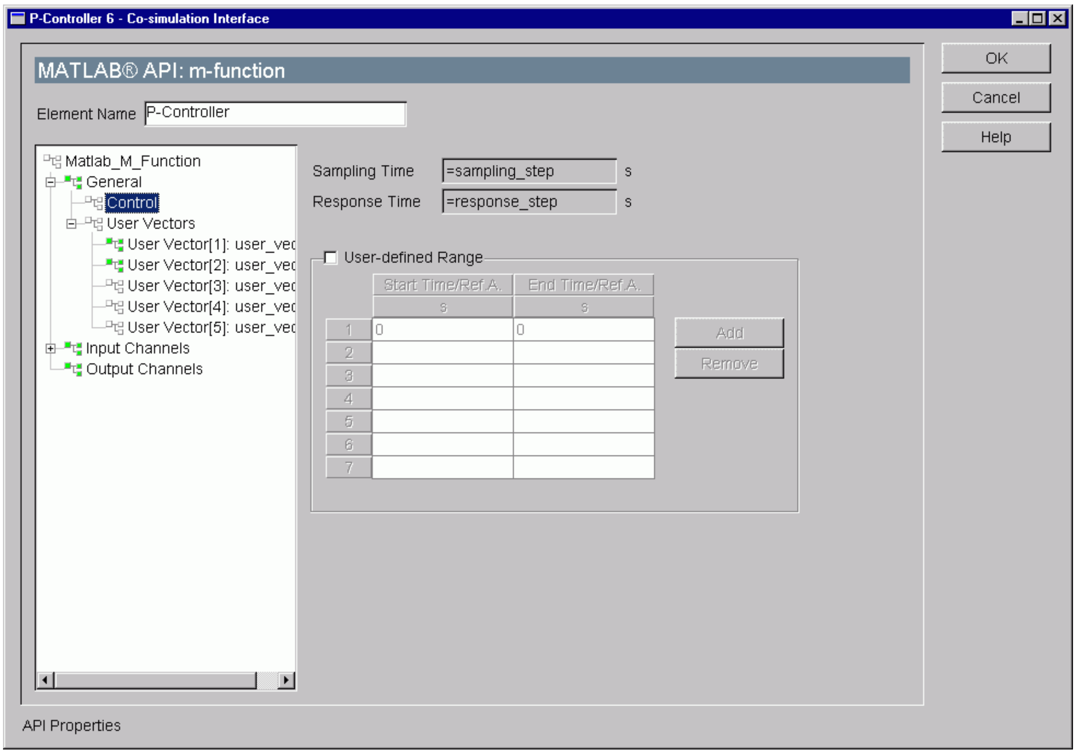


Figure 8-28: Input Dialog Box of P-Controller

Define a user vector by click on the `User Vector[1]` in the dialog element tree (refer to Section 8.1.1.3 for creating the user vectors). As already mentioned, two user vectors `Difference` and `Diameter` are used in our example:

`Difference: 0, 0.1, 0.25, 0.5, 0.75, 1`

`Diameter: 0, 0.0005, 0.0008, 0.001, 0.0012, 0.0013`

Now open Input and Output Channels dialog boxes to define input and output channels. Specify two input and one output channels (refer to *Section 8.1.1.4* and *Section 8.1.1.5* for selecting input and output channels, respectively).

Select `External Value` in the first input channel and assign it to global parameter `max_pressure` in **Value** column. Set its value to 1250e5 in **Parameters** dialog. In the second input channel select pressure in Controlled Volume as follows: Controlled Volume (3) in **Element** column, El. output parameters in **Type of Property** column and pressure in volume [Pa] in **Property** column.

In Output channel, select the diameter of Control Throttle in the following way: Control Throttle (4) in **Element** column, El. input parameters in **Type of Property** column and throttle cross-section diameter [m] in **Property** column).

Save the model under the name `pressure_control_m_1.hyd`.

8.2.3. Output Parameters

Dialog Output Parameters of **MATLAB® m-function** element is the same as of **MATLAB® Simulink** element (refer to *Section 8.1.3*).

Select the first and second input and first output channel as shown in Figure 8-29.

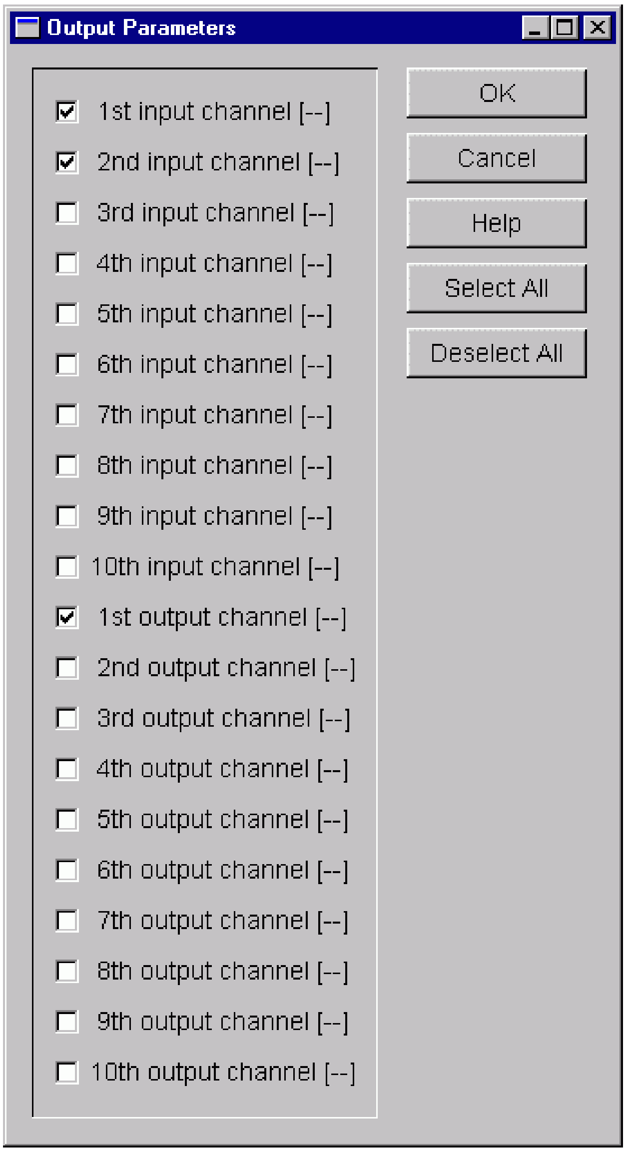


Figure 8-29: Output Data of MATLAB m-function Element

8.2.4. Running the Calculation

There are no **Initial Conditions** in MATLAB® API: m-function element.

Perform the calculation with all three cases (refer to *Section 4.4*).

Note that during calculation the plot of m-function output results appears on the screen as shown in Figure 8-30. It depicts the actual value of the effective diameter of Control Throttle after each 50th calculation step.

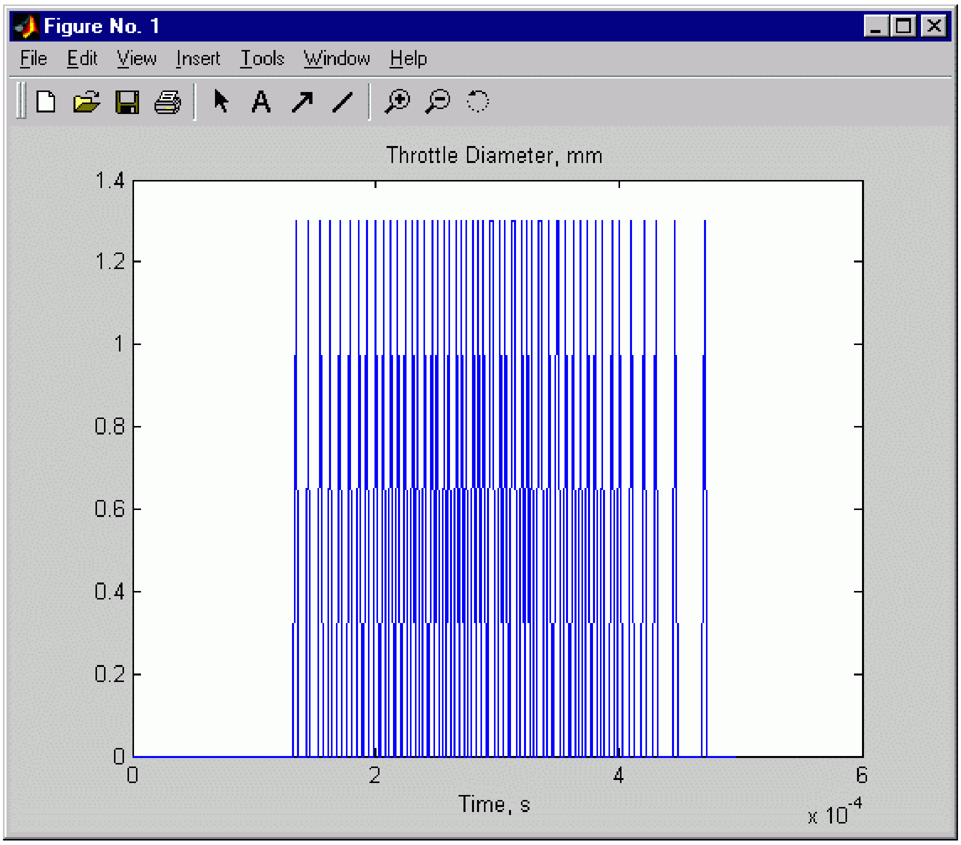


Figure 8-30: Output Results of m-function at Co-simulation

8.2.5. Calculation Results

Figure 8-31 shows the calculation results for all three cases. Upper graph depicts the pre-defined boundary pressure in Input Pressure element and calculated pressures in Controlled Volume element. Middle graph shows the cumulative flow rates through Control Throttle resulting from the P-Controller performance. Bottom graph displays the input parameters of P-Controller: rated (limiting) pressure and actual pressure. The 1st case has the smallest sampling and response steps (5e-7 and 2e-7 s, respectively). Naturally it appears to be the best theoretical controller because it regulates the pressure very smoothly. However, such fast controller is practically impossible to realize. The 2nd case has larger sampling and response steps (1e-6 and 4e-7 s, respectively). The control efficiency in this case is still very good but the pressure trace becomes somewhat noisy. The 3rd case has more realistic sampling and response steps (2e-6 and 5e-7 s, respectively). In this case the controller reacts much slower to the pressure changes, therefore it cannot prevent short pressure excursions above the limit (1250 bar). Nevertheless, even in this case the controller operation is satisfactory.

Note that further increase of sampling and response steps will lead to the situation when the controller fails, i.e. it is not capable anymore to keep the pressure under the limiting value. This we leave for the user as an exercise.

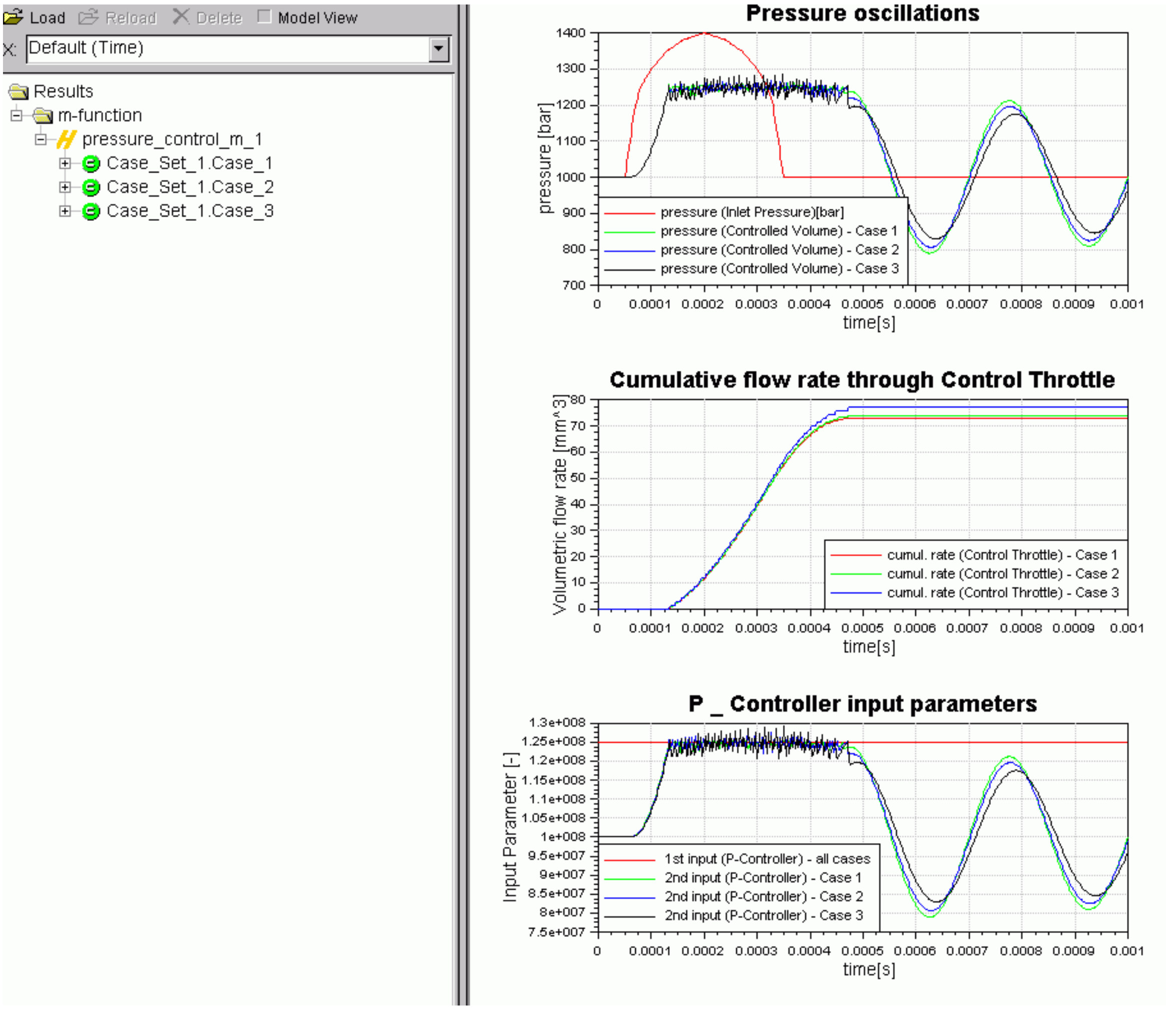


Figure 8-31: Calculation Results for Different P-controller Parameters (3 cases)

8.3. Example with MATLAB® DLL

8.3.1. Overview

In this chapter, BOOST Hydsim example with a MATLAB® DLL element will be shown. As a basis for it, the model of the common rail injector from *Chapter 5.1* is used. MATLAB® DLL element will be a Hydsim applied for the modeling of a PID Controller. The controller is aimed to suppress pressure oscillations in Branch Volume caused by injection process. These oscillations are clearly visible in *Figure 5-17*. In this case, the MATLAB® DLL element formally can be treated as a ‘variable orifice’ switched in parallel with Injection Tube. This ‘variable orifice’ has to generate additional flow rate between Rail and Branch Volume in the right direction.

The layout of the injector is shown in *Figure 8-32*. The model consists of the same elements as the model in the *Chapter 5.1*, only one new element (No. 20[10]) - PID Controller (DLL) - is added. PID Controller regulates pressure in Branch Volume according to the pressure difference between Rail Pressure and Branch Volume.

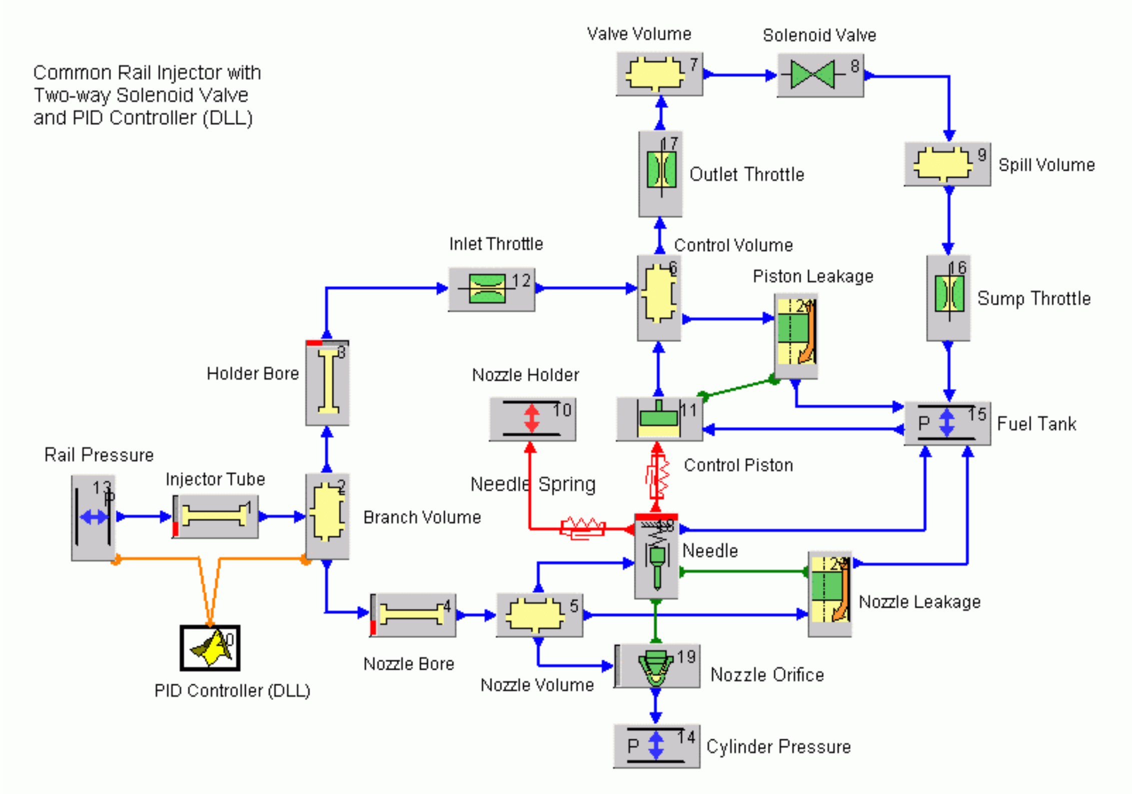


Figure 8-32: BOOST Hydsim Model with MATLAB® DLL Element

8.3.2. Input Data

Figure 8-33 shows the input dialog box of PID Controller (DLL).

[10] Numbers given for elements may differ from those in your model. Reference Figure 8-32 for numbered items.

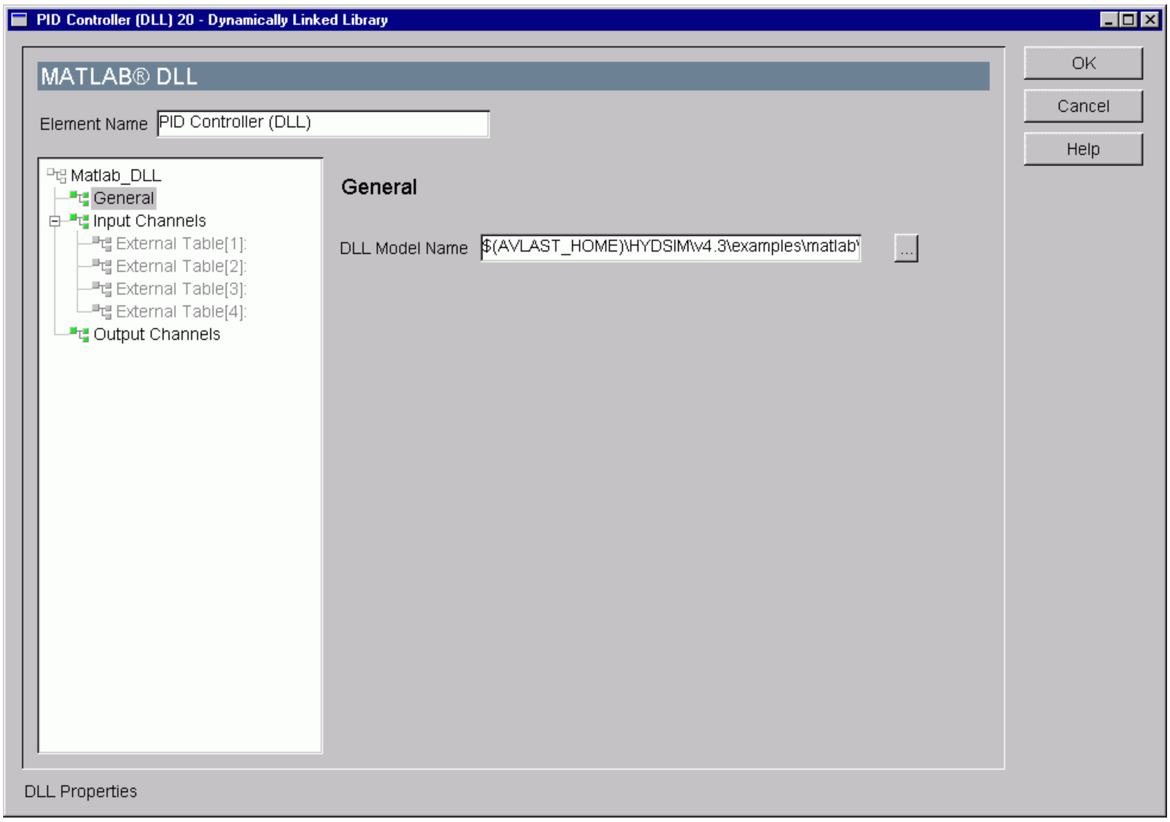


Figure 8-33: Input Dialog Box of PID Controller (MATLAB® DLL)

General part of input dialog of MATLAB® DLL element contains a single input: full path of a dll file. Input and Output Channel dialog boxes are completely same as those of MATLAB® API (Simulink and m-function) elements.

Contrary to MATLAB® API elements, MATLAB® DLL element has no Control and User vector input dialogs.

Refer to *Section 8.3.3* for creating a MATLAB® dll file.

For our model, define Input Channels as follows:

1st: Element, Rail Pressure (13), El. output parameters, pressure [Pa]

2nd: Element, Branch Volume (2), El. output parameters, pressure in volume [Pa]

3rd: Element, Branch Volume (2), El. local fluid properties, bulk modulus [N/m^2]

4th: Element, Branch Volume (2), El. output parameters, actual volume [m^3]

Next, specify one Output Channel:

- Branch Volume (2), El. output parameters, additional flow rate [m^3/s]

Set `Time step` in **Calculation Control** dialog to 2e-7 s `and Calculation end time` to 0.008 s.

8.3.3. Creating MATLAB® DLL file

Creating of MATLAB® DLL file requires the installation of SIMULINK™ and Real-Time Workshop™ tools of MATLAB® software package.

Refer to the *Section 8.1.2* for creating the Simulink model of the PID controller. In our case it calculates additional flow rate from the continuity equation:

$$\dot{q} = \frac{\Delta p \cdot V}{E},$$

where:

$\dot{q}$ .... additional flow rate

Δp .... pressure difference

V .... actual volume of Branch Volume

E .... bulk modulus of fluid

PID components are calculated according to the `Ziegler Nichols tuning method: open loop reaction rate` (refer to *Section* 8.1.2). Assuming pressure drop of 50 bar and reaction time of 1.6e-8 min we obtain the following values:

$$K_p = 32,$$

$$K_I = 8 \cdot 10^9,$$

$$K_D = 3.2 \cdot 10^{-8}.$$

Simulink model of the PID controller is shown in *Figure 8-34*.

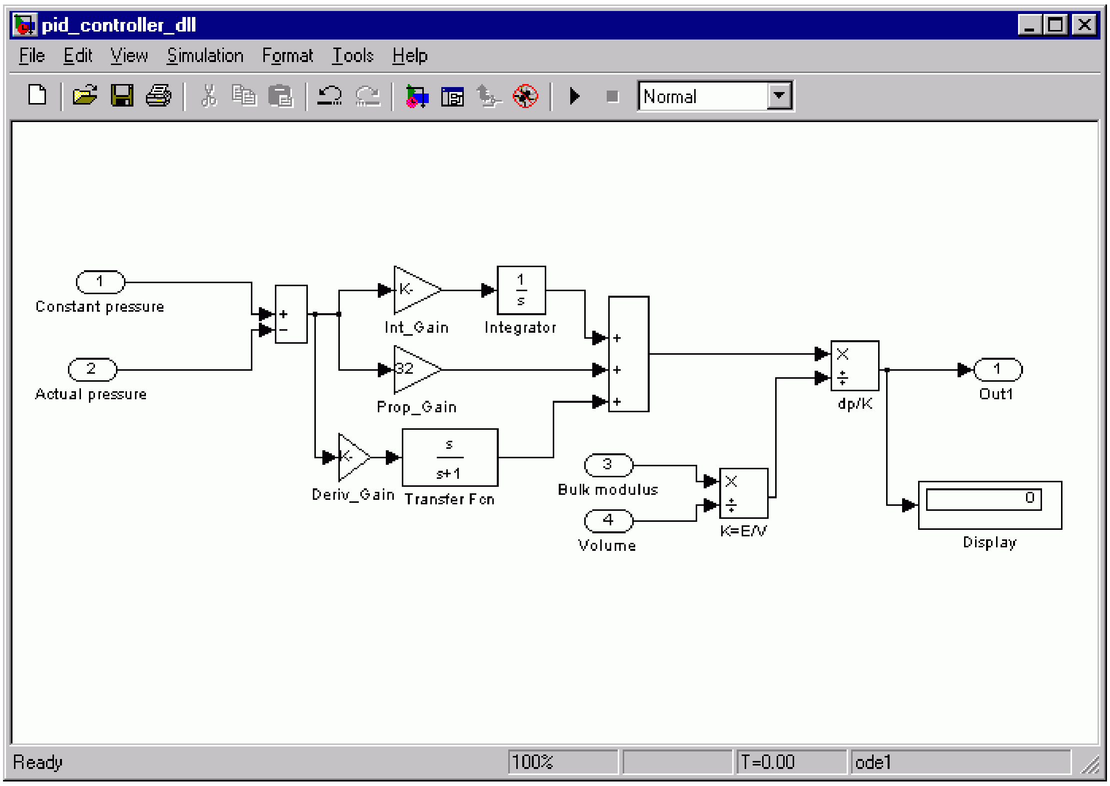


Figure 8-34: Simulink Model of PID Controller (DLL)

The following procedure may be used for generating of the DLL by MATLAB® Real Time Workshop software. This procedure works with MATLAB® versions v5.3, v6.0, v6.1, v6.5 and higher.

By typing in `mex -setup` in the MATLAB® command line, a menu pops up where the c++ compiler for DLL generation has to be selected.

Depending on the MATLAB® version, the following path has to be added to the MATLAB®/Simulink path. It is necessary to point to the `avl_grt_dll_nt.tmf` (or `avl_grt_dll_unix.tmf`) and `avl_grt_dll.c` files:

MATLAB® v5.3: `BOOST Hydsim_HOME.. \ .. \matlab\v5.3`

executing the MATLAB® command '`addpath`'('path_to_add')**Path**

MATLAB® v6.0 and higher: `BOOST Hydsim_HOME.. \ .. \matlab\v6.x`

via MATLAB® GUI (**File | Set Path** select **Add Folder Path** and **SAVE**)

> **Note:** For a DLL generation, use file avl_grt_dll_nt.tmf from the appropriate subdirectory (v5.3, v6.x) depending on MATLAB® version.
>
> **Note:** It is useful to define an environmental variable $(BOOST Hydsim_HOME) pointing to the location directory of BOOST Hydsim executable BOOST Hydsim.exe, i.e. $(AVLAST_HOME)\BOOST Hydsim\v4.5\bin\$PLATFORM.

Open **Solver** window from **Simulation|Simulation Parameters** PullDown menu and set its parameters as shown in *Figure 8-35*.

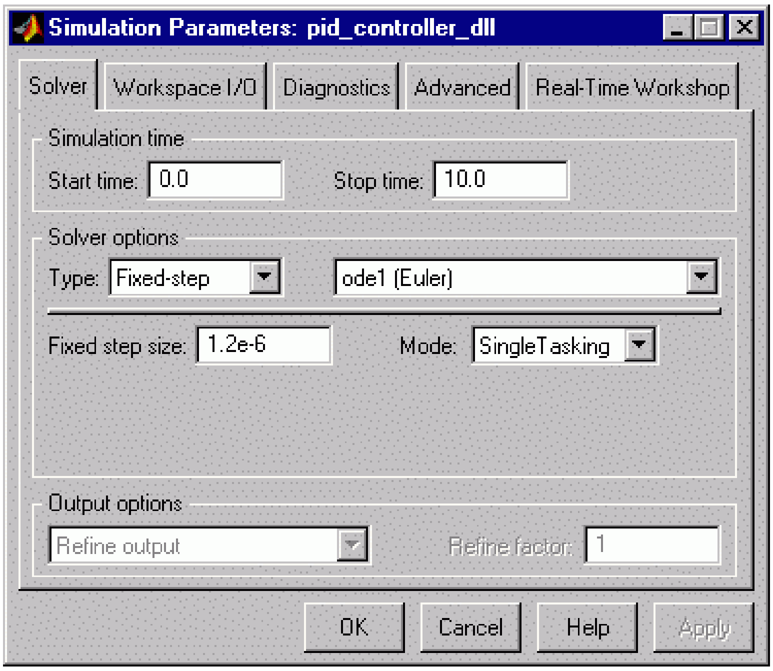


Figure 8-35: SIMULINK Solver Settings

In the **Solver** window, the incrementation and Solver type have to be defined. The incrementation must be constant (Fixed-step) and adjusted to the BOOST Hydsim time step (`2e-7` s for `Time step` in the **Calculation Control**). BOOST Hydsim will reference this parameter to its own time step, i.e. send input parameters to the DLL model and receive results back after the DLL incrementation time. This principle is similar to the response time in MATLAB® API element (refer to *Section 8.1.1.2*).

☞ **Note:** The incrementation must be greater than or equal to the BOOST Hydsim time step and smaller than BOOST Hydsim calculation end time. Otherwise, program will issue an error message.

The values for **Start** and **Stop Time** have no meaning because BOOST Hydsim will reset them during calculation.

☞ **Note:** If no integrators, memory blocks etc. are used in the Simulink model, the value of the Fixed-step size will be ignored and automatically set to the BOOST Hydsim time step. If DLL incrementation is not a multiple of the BOOST Hydsim time step, program will use the first higher multiple as DLL incrementation.

In the **Workspace I/O** window shown in *Figure 8-36*, **Input** and **Initial state** parameters must be inactive.

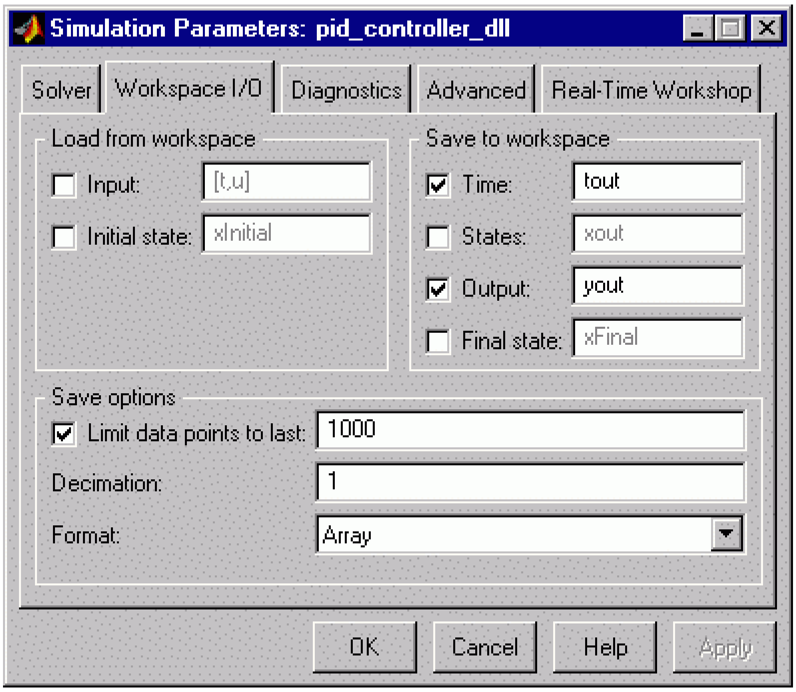


Figure 8-36: SIMULINK Workspace Settings

In the **Real-Time Workshop** window, shown in *Figure 8-37*, the settings for the DLL generation have to be defined.

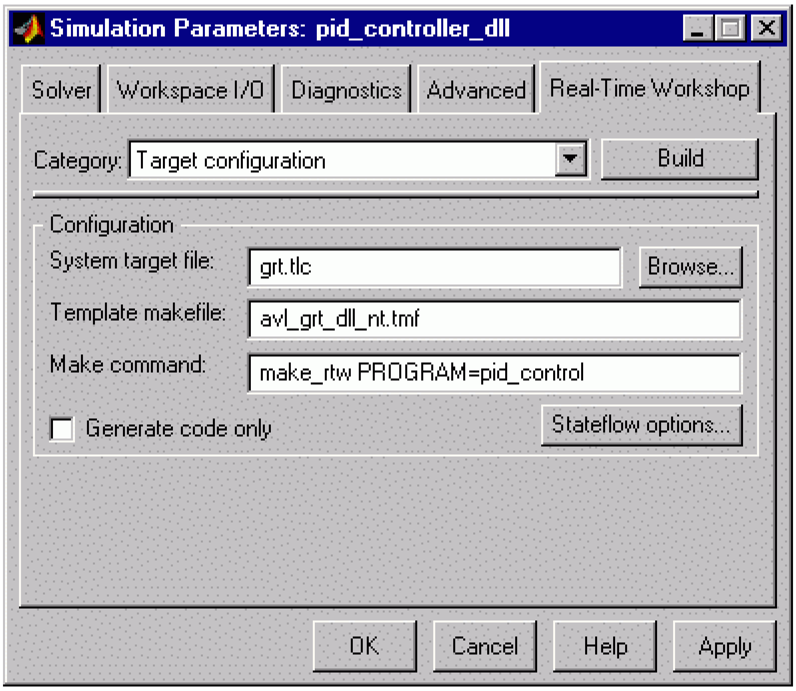


Figure 8-37: SIMULINK Real-Time Workshop Settings

System target file window defines the type of the code which can be produced. Enter `grt.tlc` (general real-time).

Template makefile window defines the name of the template file from which the new `Makefile` is generated. For the generating of the desired DLL file using the C++ V6.0 compiler on Windows, enter the template file `avl_grt_dll_nt.tmf`. On Linux/Unix platforms, enter the template file `avl_grt_dll_unix.tmf`. This file and `avl_grt_dll.c` file must be stored either in the directory `%MATLAB ROOT%\rtw\c\grt` or in the current working directory (if **Set Path** is not specified). Otherwise the complete path name must be entered.

Enter `make_rtw PROGRAM=pid_control` in **Make command** field. `PROGRAM=pid_control` defines the file name of the DLL. Alternatively the model name is used and an underscore is placed in front of it. If the DLL file name has to be same as the MDL file (Simulink model) name, an underscore must be placed in front (refer to the table below). Otherwise no valid C-MEX DLL will be generated, as the MDL file in MATLAB® could be not opened any longer, because MATLAB® always checks first whether a DLL file with the same name is existing or not.

Make command	**Simulink model**	**Created DLL**
`make_rtw`	`example.mdl`	`_example.dll`
`make_rtw PROGRAM=example`	`example.mdl`	`_example.dll`
`make_rtw PROGRAM=test`	`example.mdl`	`Test.dll`

Click **Build** button to generate a C-code. The `Makefile` will be produced, executable `nmake.exe` will be called from this `Makefile`, and the object files will be linked to the DLL.

☞ **Note:** For MATLAB® v6.0 and higher, the DLL and other necessary files will be created in the new directory named model_name_grt_rtw. This directory will be placed in the current working directory.

8.3.4. Output Parameters

BOOST Hydsim supports the post-processing of all input and output channel parameters of MATLAB® DLL element and DLL incrementation. To perform this, select desired output parameters from the **Element | Store Results** menu. These output results may be used to control the interaction between BOOST Hydsim and MATLAB® DLL.

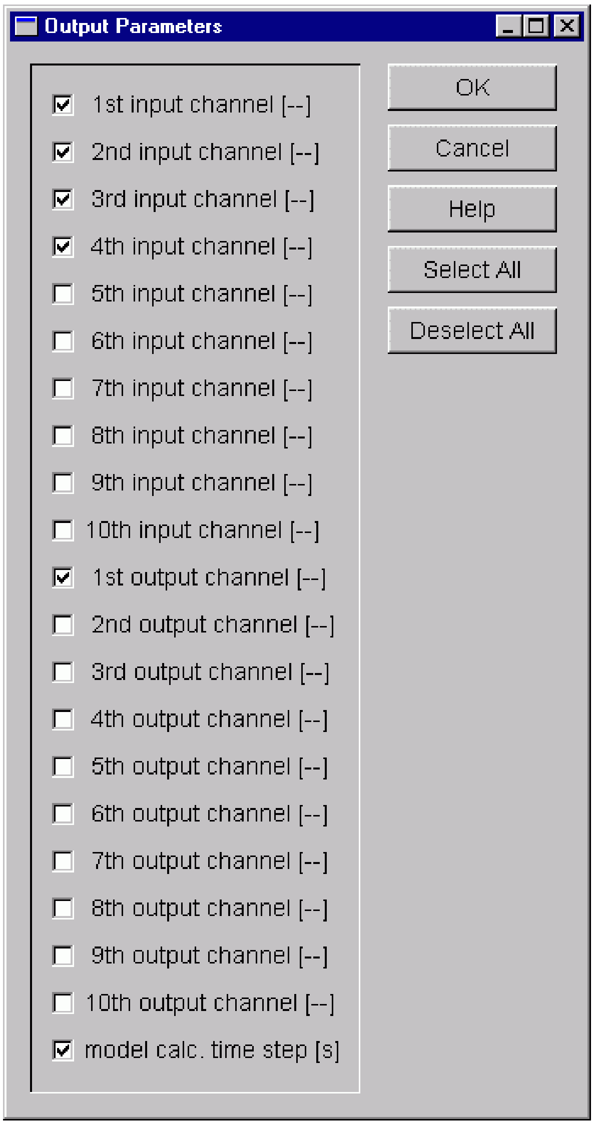


Figure 8-38: Output Data of MATLAB DLL Element

8.3.5. Running the Calculation

For performing the calculation, refer to *Section 4.4*.

8.3.6. Calculation Results

To view the calculation results, select **Show Results** from **Simulation** Pulldown menu. The new Impress Chart window with the **Result** tree of the active BOOST Hydsim model will pop up. Next, load the results of the original model `common_rail2wv.hyd` from *Chapter 5.1* for comparison.

Figure 8-39, upper graph shows the pressure in Branch Volume, for the models with and without controller, respectively. Obviously, the model with the PID Controller exhibits very small pressure oscillations compared to the original model. Middle and bottom graphs depict the parameters of Input and Output Channels of the controller, respectively.

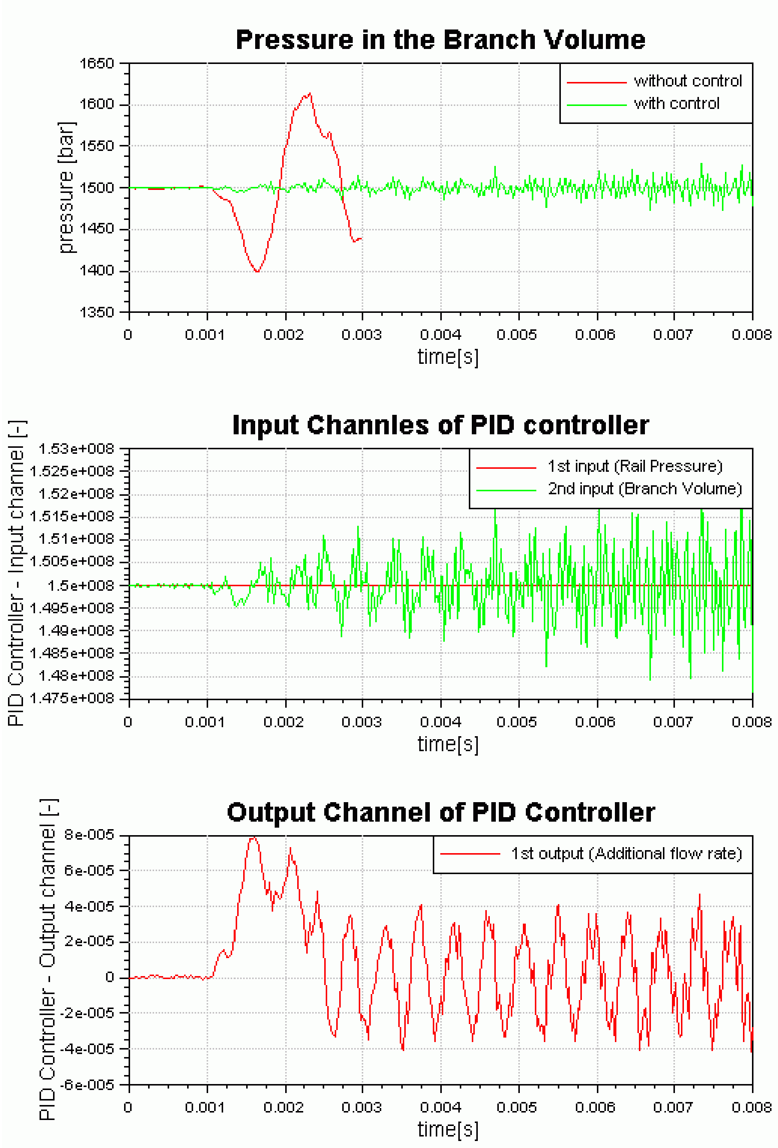


Figure 8-39: Pressure in Branch Volume and PID Controller Data (DLL)

Figure 8-40 shows the comparison of the pressure in Nozzle Volume, Needle motion and injection rate. In the model with PID Controller, the pressure oscillations in Nozzle Volume are considerably smaller. The needle lift and injection rate remain very close to the original model results. The user should not search for deep physical meaning in these results. It is not a model of a real control system but just a demonstration example.

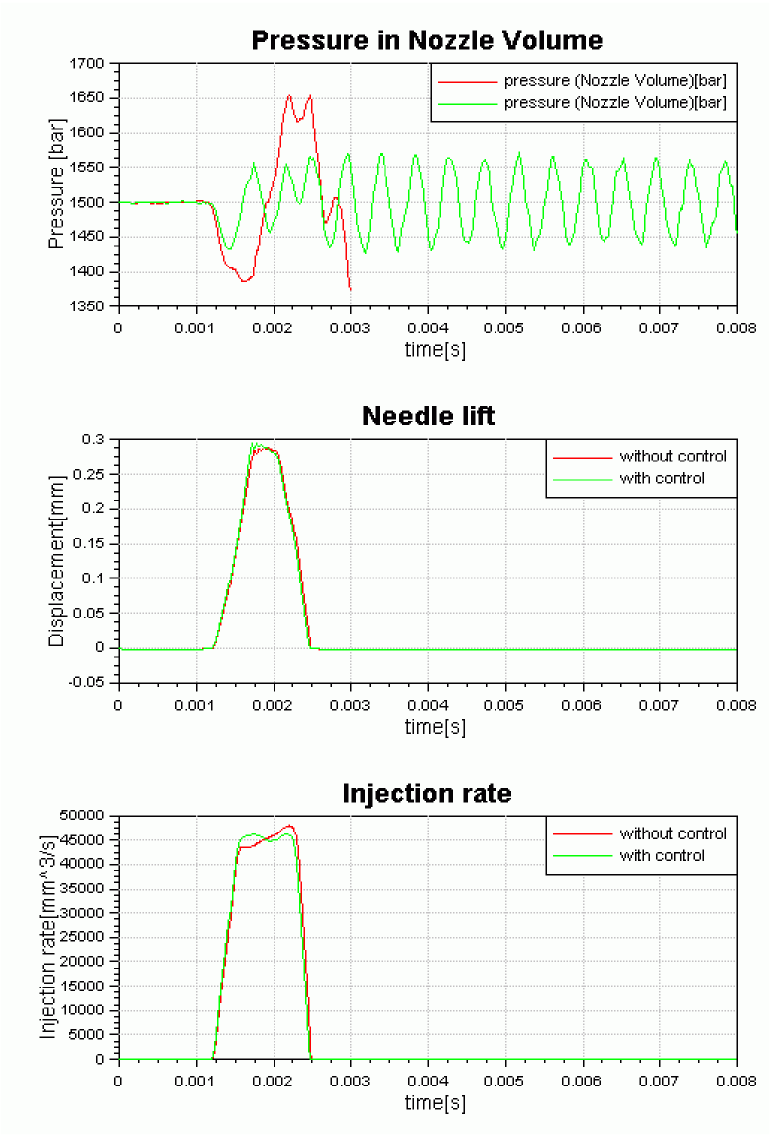


Figure 8-40: Pressure in Nozzle Volume, Needle Lift and Injection Rate

9. BOOST HYDSIM – BOOST LINK

9.1. Introduction

BOOST-Hydsim-BOOST Link is used for the co-simulation of liquid-gaseous (hydraulic-pneumatic), mechanical-gaseous or combined hydromechanical-gaseous systems by BOOST Hydsim and BOOST Cycle Simulation. There are two BOOST link elements in **External** group of BOOST Hydsim element tree: **Boost Link (Plenum)** and **Boost Link (Valve)**. **Boost Link (Plenum)** element in BOOST Hydsim represents the interface to the **Plenum** element in BOOST while **Boost Link (Valve)** element represents the interface to the **Check Valve** element in BOOST. Both link elements have to be connected to a **Standard Piston** element. Data exchange between BOOST Hydsim and BOOST Cycle simulation kernels is carried out via ACCI interface.

A typical application example of BOOST Hydsim-BOOST co-simulation is the gas injector (or gas valve) controlled by a hydraulic fluid. The principal scheme of such injector is shown in Figure 9-1. The injector consists of the gas metering valve, needle with spring, outwards opening nozzle and hydraulic control system. Gas elements are depicted in brown color, liquid elements – in cyan color. The needle, metering valve body and control liquid part is simulated by BOOST Hydsim while entire gas part is modeled in BOOST. Figure 9-2 shows the division of the injector into BOOST and BOOST Hydsim parts for the further sub-model creation.

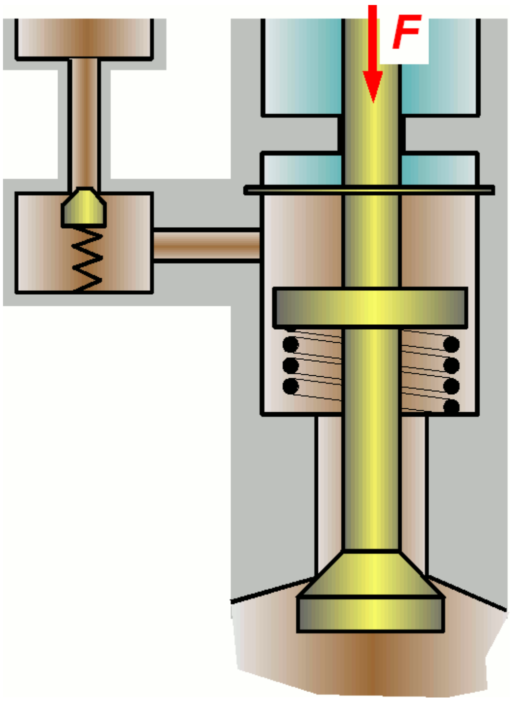


Figure 9-1: Schematic of outwards opening gas valve with hydraulic control

Model of the gas injector (valve) consists of three different types of physical elements:

- gas elements (plenums, pipes and throttle)
- hydraulic elements (volumes and orifices)

- mechanical elements (needle, piston and springs)

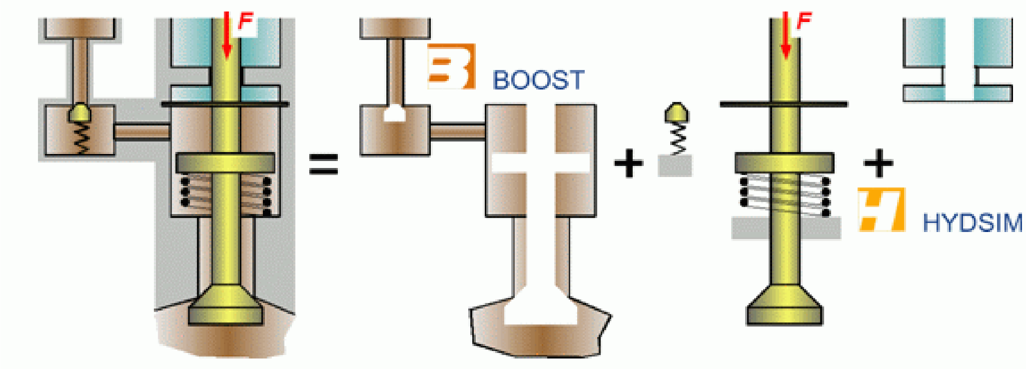


Figure 9-2: Gas injector division into BOOST and BOOST Hydsim sub-models

As stated earlier, BOOST Hydsim covers the simulation not only of the control liquid flow but also the motion of mechanical elements (multi-body dynamics). Interface between BOOST Hydsim and BOOST simulation is based on the **Standard Piston** element from BOOST Hydsim side and **Plenum** or **Check Valve** element from BOOST side. Gas pressures in the gas system are calculated by BOOST. These are typically stagnation pressures in **Plenum** elements or dynamic pressures upstream and downstream of **Check Valve** elements. **Piston** areas pressurized by gas are defined and controlled in BOOST Hydsim sub-model.

For each piston-type element BOOST Hydsim calculates coordinate and velocity. For a **Piston** element connected to a **Boost Link (Plenum)** element, the product of piston velocity and pressurized area yields the flow rate (called also displacement flow). This displacement flow is transferred to BOOST **Plenum** element at every data exchange step. For a **Piston** element connected to a **Boost Link (Valve)** element, piston input/output coordinate and velocity are transmitted to BOOST **Check Valve** element. In return, BOOST transfers to BOOST Hydsim the gas pressures and their gradients acting on the respective **Piston** elements. The data exchange between BOOST Hydsim and BOOST at the needle valve seat is illustrated in Figure 9-3.

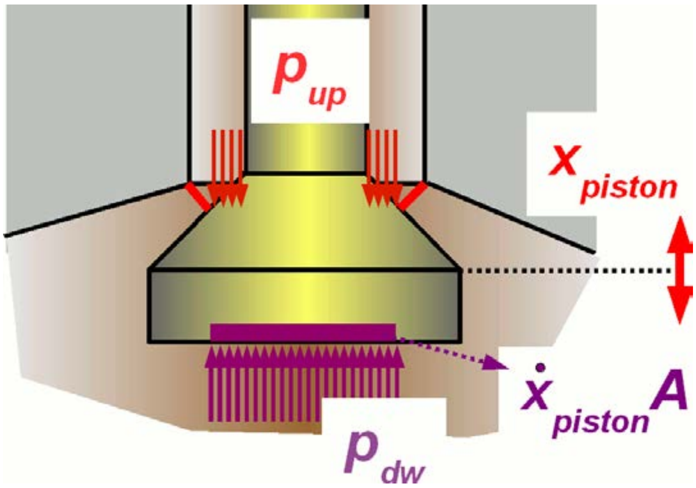


Figure 9-3: Data exchange between BOOST and BOOST Hydsim at needle seat

9.2. CNG Injector Model with BOOST Link

This chapter guides the user through the BOOST Hydsim-BOOST Link example based on the simplified model of the Compressed Natural Gas (CNG) injector. The injector schematic is same as in the previous chapter (refer to Figure 9-1), only the liquid part is omitted for simplification and the needle motion is directly controlled by a solenoid valve.

9.2.1. Creating the Boost Model

The whole gas part of the injector is modeled using BOOST software. Schematic and BOOST model of the gas system is shown Figure 9-4. It consists of the following elements:

- Supply Pressure **(SB1)**
- Metering Valve **[CV1]** */ linked to BOOST Hydsim /*
- Inlet Volume **(PL1)** */ linked to BOOST Hydsim /*
- Pipe **(3)**
- Nozzle Volume **(PL2)**
- Nozzle Valve **(CV2)** */ linked to BOOST Hydsim /*
- Cylinder Pressure **(SB2)**

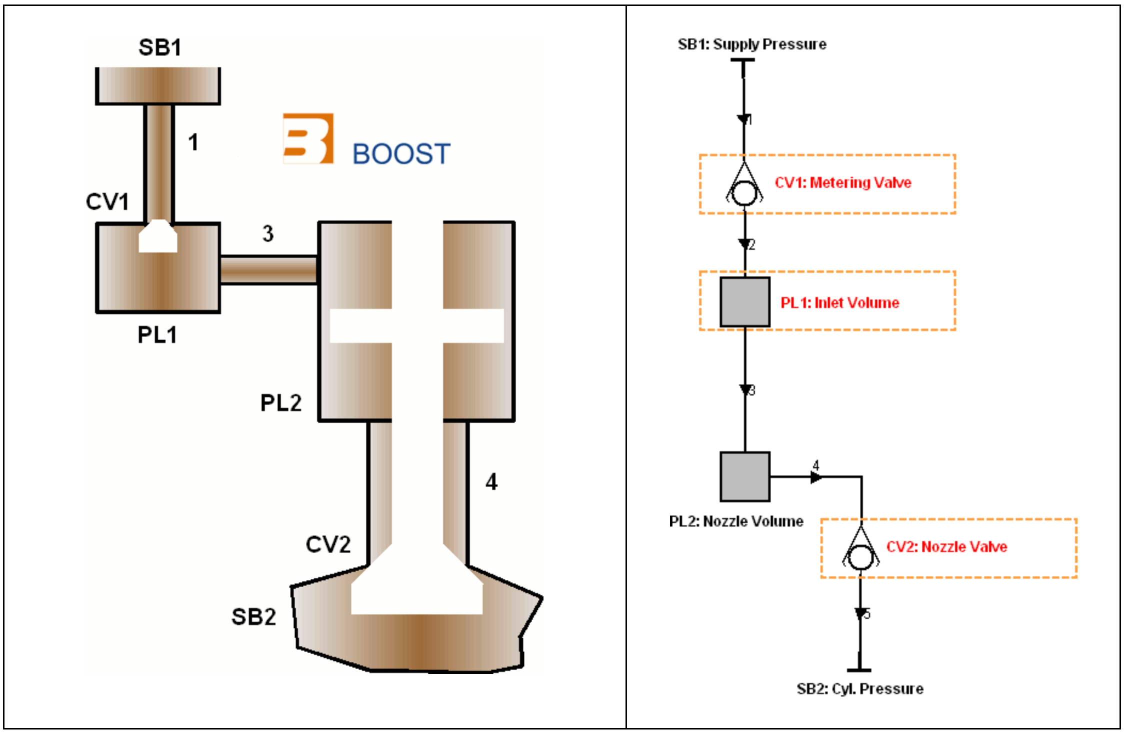


Figure 9-4: Schematic and BOOST sub-model of CNG injector

9.2.1.1. Supply Pressure

Compressed natural gas (CNG) is supplied to the injector inlet with constant pressure and temperature (system boundary SB1). Boundary SB1 is connected via pipe 1 to the metering valve CV1. Input dialog of the boundary SB1 is shown in Figure 9-5.

Supply Pressure /

Pressure: 100 bar

Temperature: 80 degC

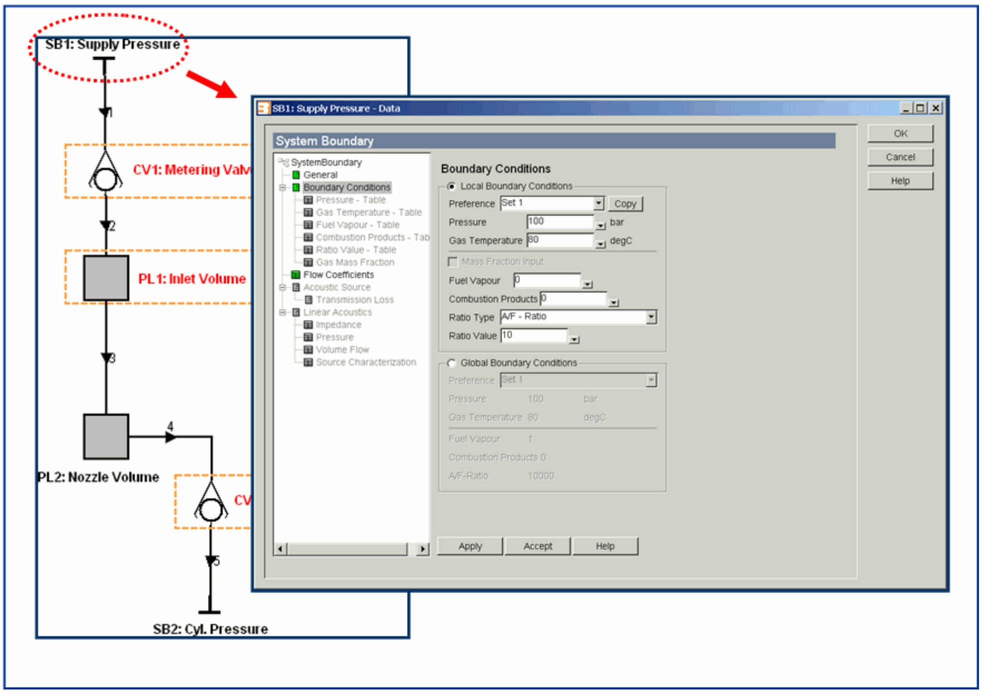


Figure 9-5: Input dialog of Supply Pressure (SB1)

Fuel vapour value is set to zero because the fuel is not a liquid but natural gas (methane).

9.2.1.2. Metering Valve

Metering Valve /

Full Model Tab: ignored – valve lift and velocity transferred from BOOST Hydsim

Flow Model Tab: linear function, at valve lift of 0.3 mm flow coefficient is 0.8

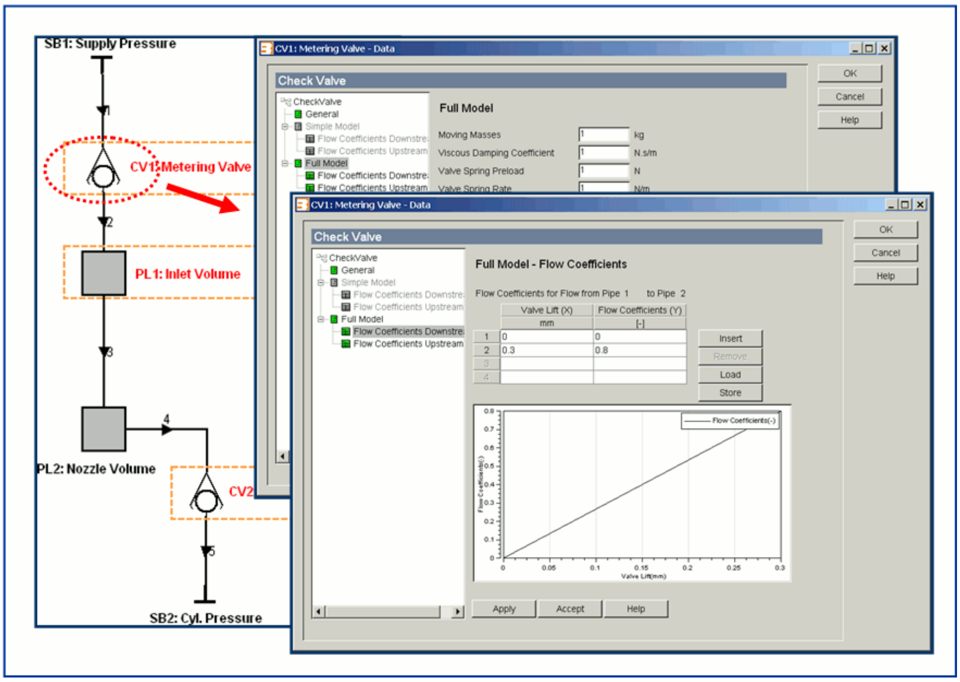


Figure 9-6: Input dialog of Metering Valve (CV1)

Next important element in BOOST sub-model is the inlet metering valve (CV1). It is just a spring-controlled mechanical valve. Its input dialog is shown in Figure 9-6. The flow characteristic of the metering valve is defined by flow coefficients as a function of the valve body lift. As Metering Valve CV1 is linked to respective BOOST Link (Valve) in BOOST Hydsim sub-model, its inertia and stiffness data will be ignored, i.e. BOOST Hydsim data will be used. This means that the valve body lift will not be calculated by BOOST but will be received from BOOST Hydsim. BOOST calculates only the gas flow through the valve including upstream and downstream pressures acting on the valve body. These pressures are transferred to BOOST Hydsim. Note that downstream flow direction in BOOST is always the flow from the pipe element with the lower number into the pipe with the higher ID number.

The list of Plenum and Check Valve elements linked to BOOST Hydsim is defined in BOOST **Control dialog** box at **User Defined Parameters** tab (refer to Chapter 9.2.1.8).

9.2.1.3. Inlet Volume

Next element is Inlet Volume-Plenum PL1. It is connected via Pipe 2 to Metering Valve CV1 and via Pipe 3 to Nozzle Volume-Plenum PL2. The input dialog of Inlet Volume is shown in Figure 9-7. This element is linked to the respective BOOST Link (Plenum) element in BOOST Hydsim sub-model. It calculated the pressure acting on the bottom area of Metering Valve body and sends it to BOOST Hydsim. In return actual it receives from BOOST Hydsim the displacement flow of the Metering Valve body.

Inlet Volume /

Volume size: 30 mm^2

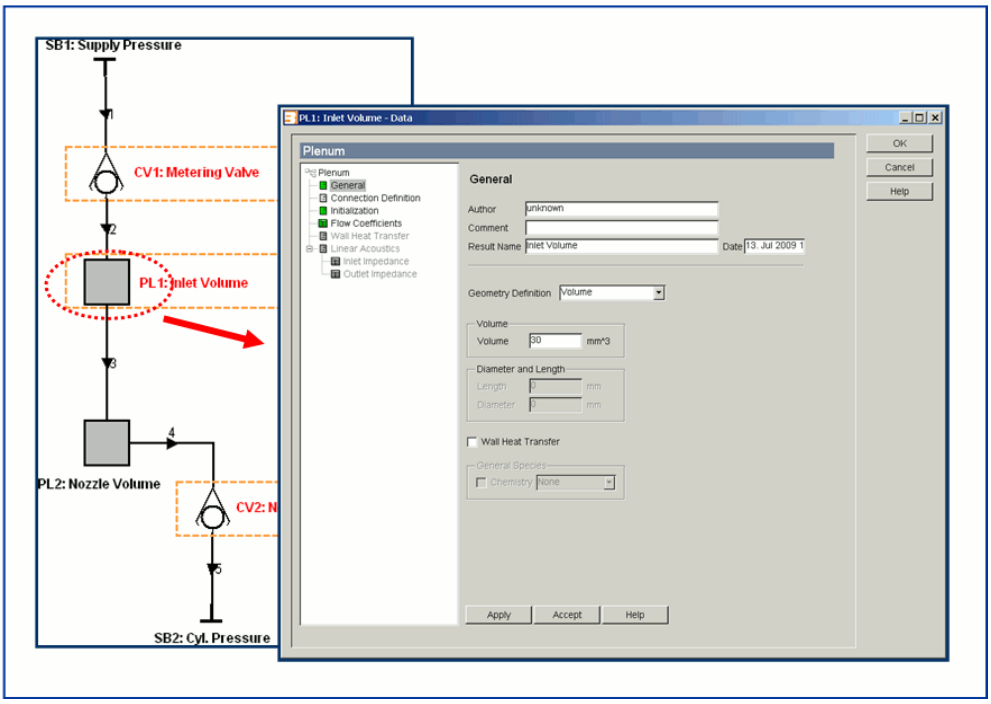


Figure 9-7: Input dialog of Inlet Volume (PL1)

9.2.1.4. Nozzle Bore

Nozzle Bore (Pipe 3) connects Inlet Volume-Plenum PL2 and Nozzle Volume-Plenum PL3. Input dialog of Nozzle Bore is shown in Figure 9-8. There laminar and turbulent friction is considered.

Nozzle Bore /

Pipe Length: 25 mm

Diameter: 1.5 mm

Laminar Friction Coefficient: 64

Friction Coefficient: 0.019

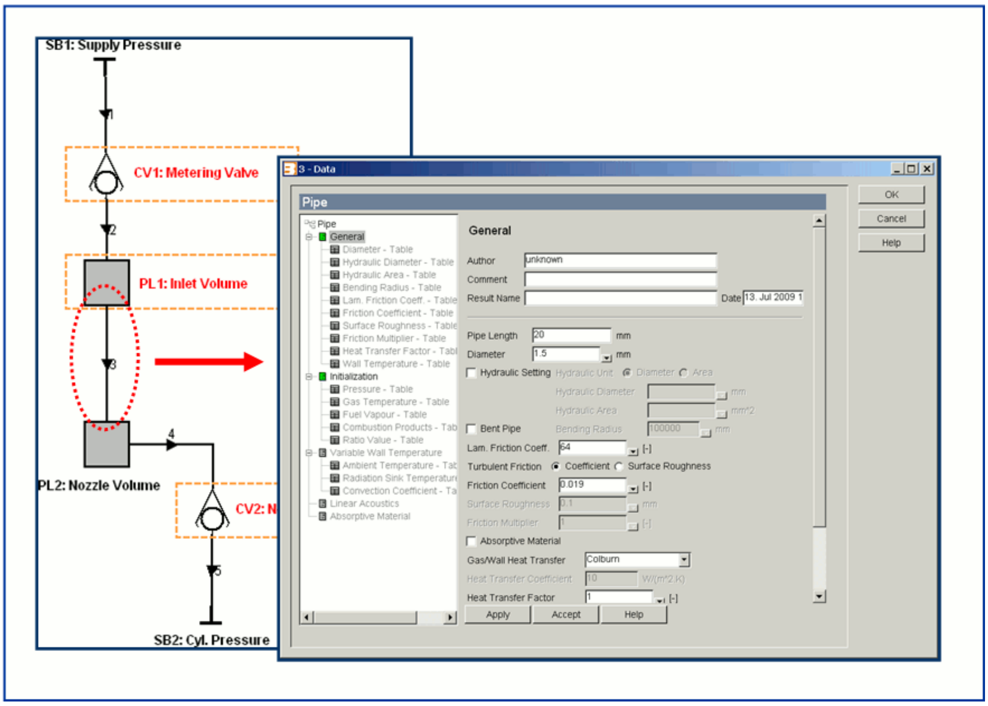


Figure 9-8: Input dialog of Nozzle Bore

9.2.1.5. Nozzle Volume

Input dialog of Nozzle Volume-Plenum PL2 is shown Figure 9-9.

Nozzle Volume /

Volume size: 30 mm^2

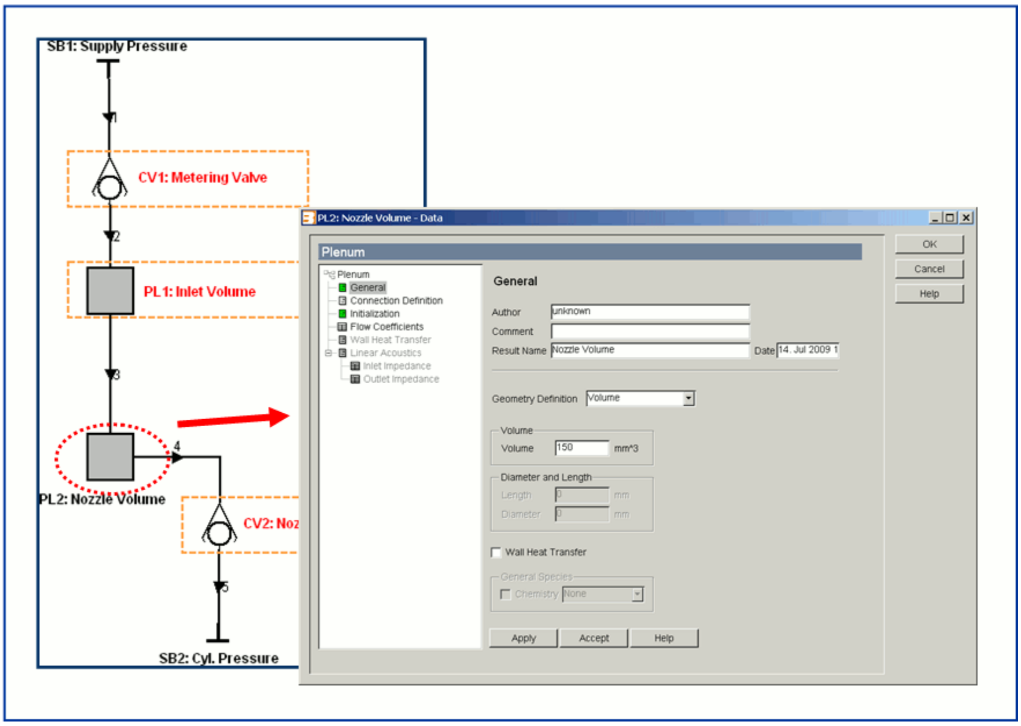


Figure 9-9: Input dialog of Nozzle Volume PL2

In our simplified injector model Nozzle Volume is simulated as one element. Volumes of all parts along the gas flow passage from Nozzle Bore to Needle Seat are summarized into one entity. For the more accurate simulation of gas dynamics inside this passage, it may be split into several volumes connected via pipes and restrictions.

9.2.1.6. Nozzle Valve

Nozzle Valve CV2 is a BOOST Check Valve element simulating the gas flow from Nozzle Volume PL2 into the cylinder. It is linked to the respective BOOST Link (Valve) element in BOOST Hydsim sub-model. Through this link the position of the valve body (Needle) is transferred to Nozzle Valve element at each data exchange step. Using this Needle lift, BOOST determines the actual values of the flow coefficients for the calculation of downstream or upstream gas flow. Input dialog of Nozzle valve is shown in Figure 9-10.

Nozzle Valve /

Full Model Tab: ignored – valve lift and velocity transferred from BOOST Hydsim

Flow Model Tab: linear function, at valve lift of 0.3 mm flow coefficient is 0.8

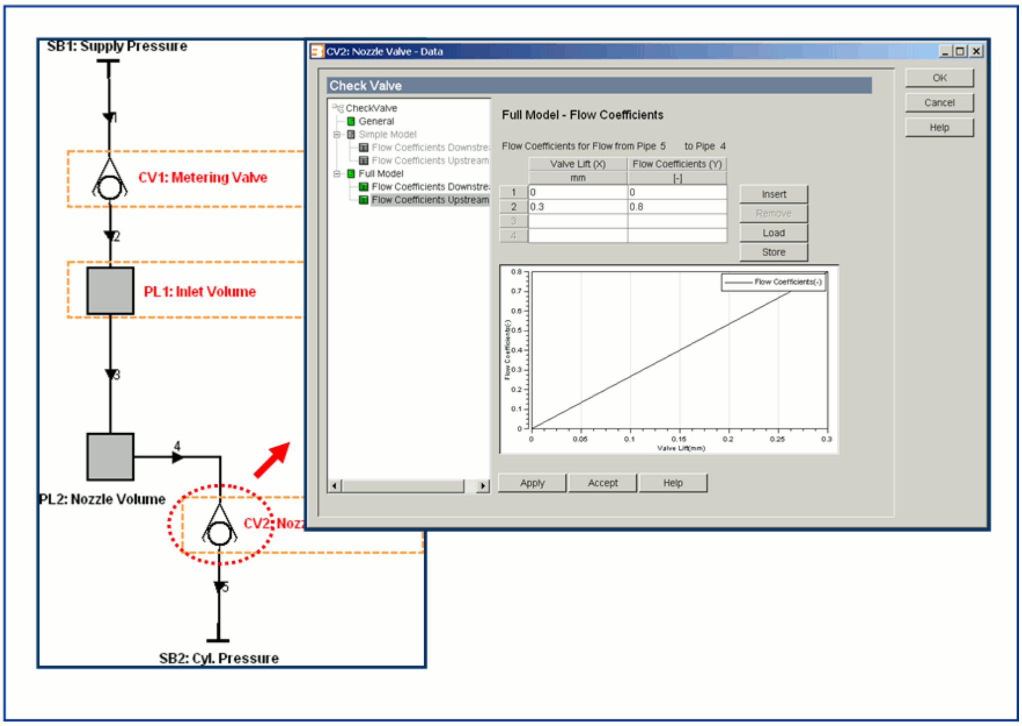


Figure 9-10: Input dialog of Nozzle Valve CV2

9.2.1.7. Cylinder Pressure

The last element of the BOOST sub-model is the Cylinder Pressure boundary SB2. Input dialog of this boundary is shown in Figure 9-11. There pressure and temperature in the cylinder for simplicity are assumed to be constant. This is justifiable over a short injection event (24 deg CA). In a real cylinder chamber pressure and temperature may vary, of course.

Cylinder Pressure /

Pressure: 10 bar

Temperature: 100 degC

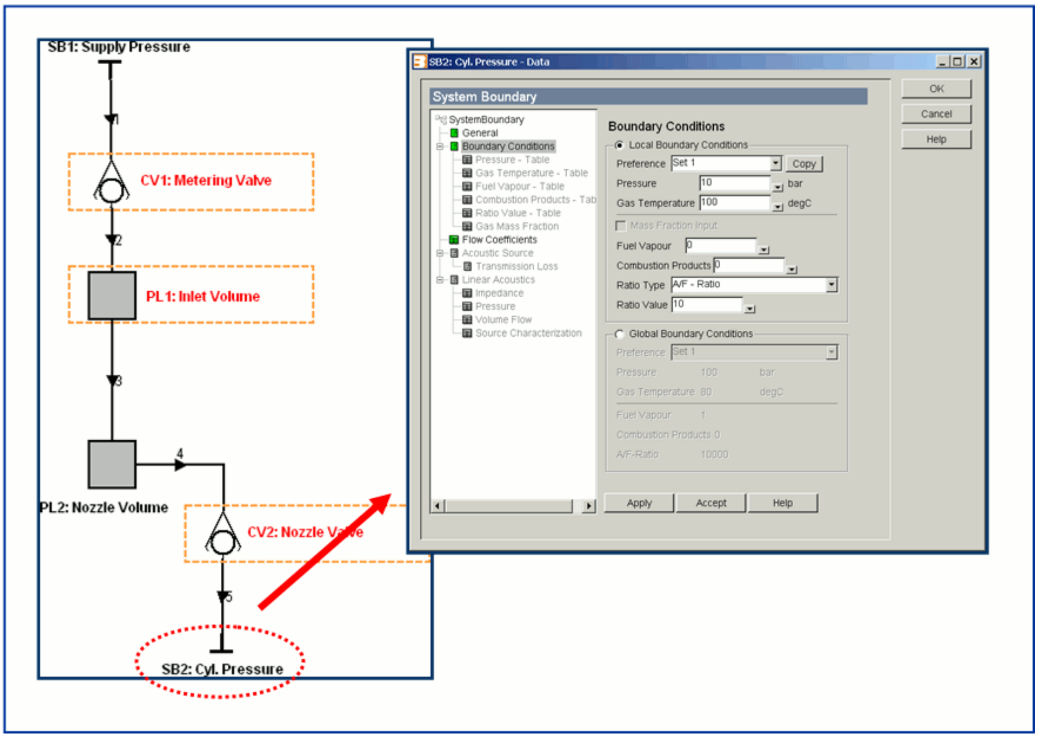


Figure 9-11: Input dialog of Cylinder Pressure boundary SB2

9.2.1.8. User Defined Parameters

As stated earlier, three elements in the BOOST sub-model (Metering Valve CV1, Inlet Volume PL1 and Nozzle Valve CV2) are linked with their respective counterparts in the BOOST Hydsim sub-model. These links in BOOST are specified via **User Defined Parameters** under **Simulation | Control** dialog shown in Figure 9-12. There one has to define whether the co-simulation with BOOST Hydsim is active or not (ON/OFF), specify the name of host computer for BOOST Hydsim co-simulation (e.g. localhost), port number for ACCI interface (e.g. 7777) and names of elements which are constrained with the communication channels (CHANNEL1= CHECKVALVE*1, CHANNEL2= CHECKVALVE*2 and CHANNEL3= PLENUM*1).

User Defined Parameters / Simulation / Control /

1st / Parameter Key: HDSM_LNK_SIMULATION

1st / Value: ON

2nd / Parameter Key: HDSM_LNK_HOST

2nd / Value: localhost

3rd / Parameter Key: HDSM_LNK_PORT

3rd / Value: 7777

4th / Parameter Key: HDSM_LNK_CHANNEL*1

4 th / Value: CHECKVALVE*1

5 th / Parameter Key: HDSM_LNK_CHANNEL*2

5 th / Value: CHECKVALVE*2

6 th / Parameter Key: HDSM_LNK_CHANNEL*3

6 th / Value: PLENUM*1

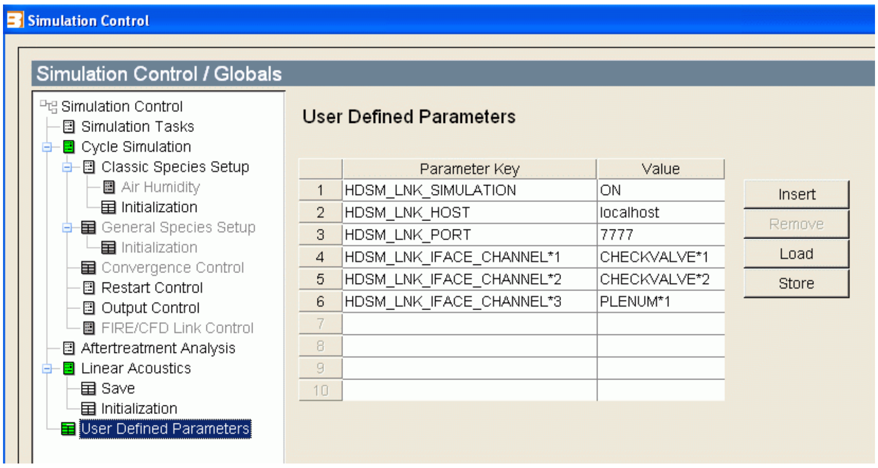


Figure 9-12: Simulation Control - User Defined Parameters Dialog

9.2.1.9. Simulation Control

Select **Cycle Simulation** in the tree to open the following window and enter the data:

Reference Speed: 2000 rpm

End of Simulation: 24 deg

Simulation Step Size: 1 deg

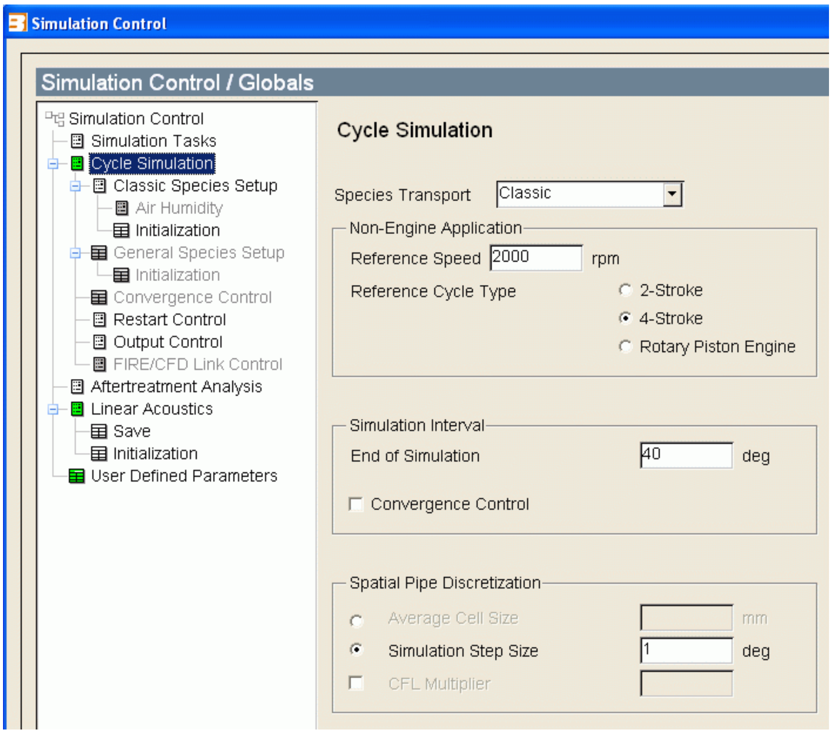


Figure 9-13: Simulation Control - Cycle Simulation Dialog

Certain control data like **Reference Speed**, **End of Simulation** and **Simulation Step Size** are defined in BOOST Hydsim which is the master software in this case. This data, if defined in the BOOST **Simulation Control** dialog, will be ignored, i.e. they will be used only as initial values.

Select **Classic Species Setup** in the tree to open the following window and enter the data shown:

Standard Fuel:	Methane
Lower Heating Value:	50000 kJ/kg
Stoichiometric A/F Ratio:	0.1 deg

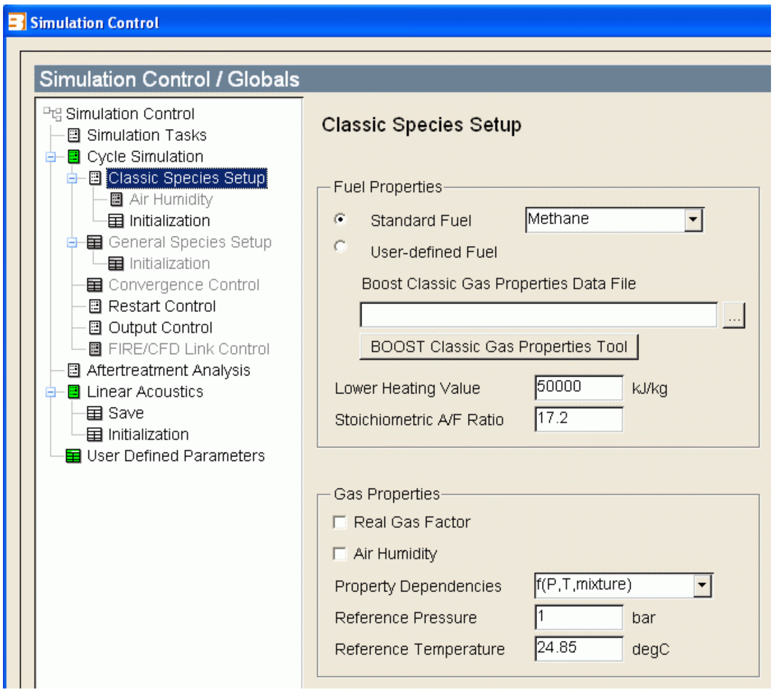


Figure 9-14: Simulation Control - Classic Species Setup Dialog

Select **Restart Control** to open the following window and enter the data shown:

Restart File Saving Interval:	Specific Interval
Saving Interval:	360 deg

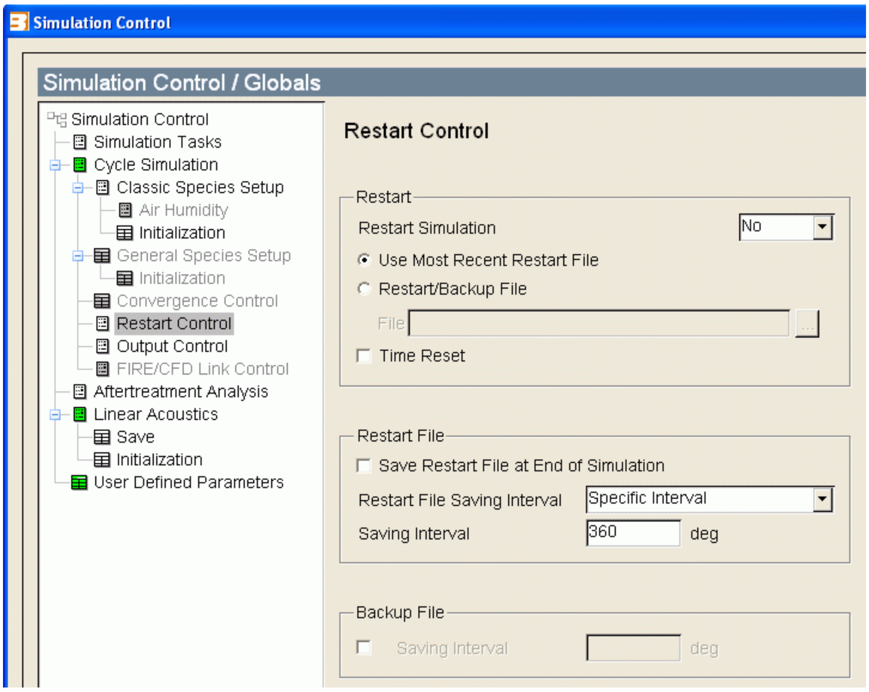


Figure 9-15: Simulation Control – Restart Control Dialog

Select **Output Control** to open the following window and enter the data shown:

Saving Interval: 0.1 deg

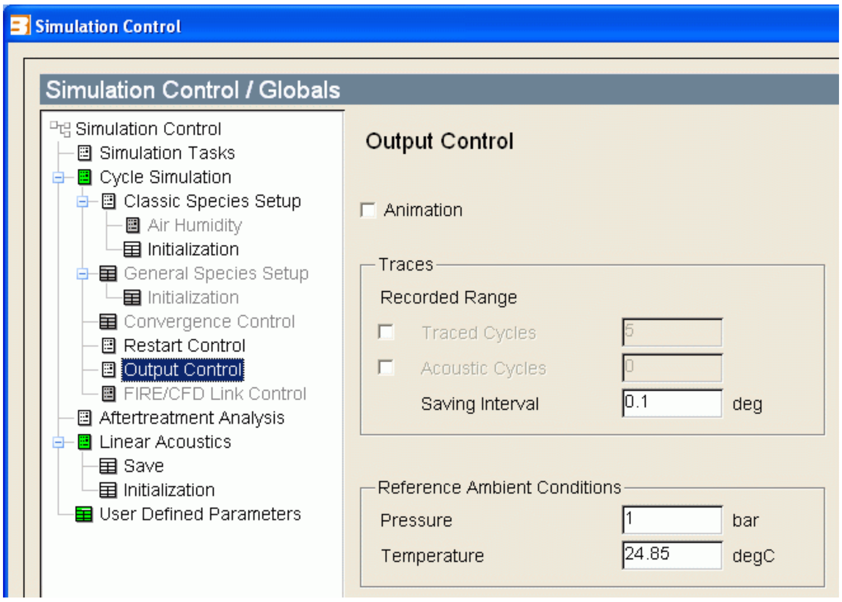


Figure 9-16: Simulation Control – Output Control Dialog

9.2.1.10. Running Boost model

BOOST calculation has to be started in a standard way by clicking **Cycle Simulation** in the **Run** dialog. In case BOOST Hydsim -BOOST co-simulation is activated in **User Defined Parameters** dialog (parameter key HDSM_LNK_SIMULATION set to ON), BOOST will automatically start ACCI server process which will ensure communication between the two kernels. The flow chart of BOOST calculation can be viewed in **Simulation Logfile** window shown in Figure 9-17.

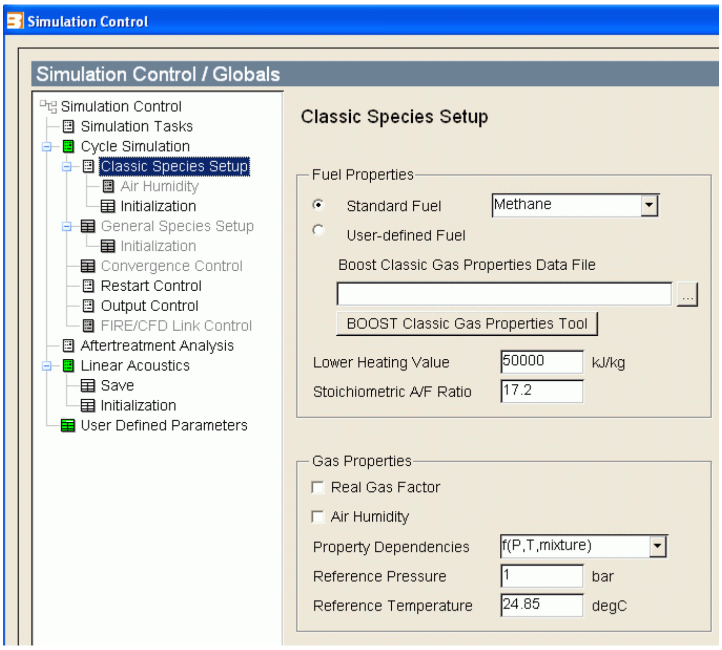


Figure 9-17: Simulation Logfile dialog

9.2.2. Creating the BOOST Hydsim Model

All mechanical injector parts of are modeled using BOOST Hydsim software. Schematic and BOOST Hydsim sub-model of the mechanical system is shown in Figure 9-18.

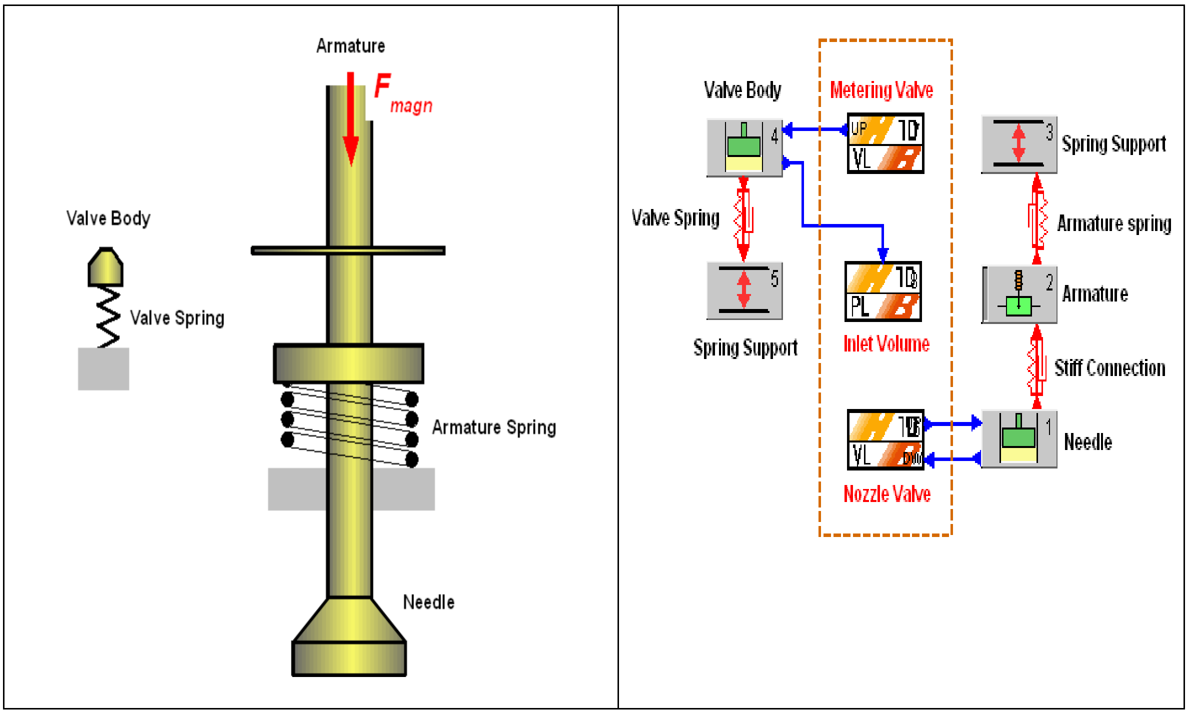


Figure 9-18: Schematic and BOOST Hydsim sub-model of CNG injector

BOOST Hydsim sub-model consists of the following elements:

- Spring Support (Valve) **(BOUNDARY/Mechanical)**
- Valve Body **[PISTON/Standard]**
- Spring Support (Armature) **(BOUNDARY/Mechanical)**
- Armature **(SOLENOID/Armature (Basic Model))**
- Needle **(PISTON/Standard)**
- Metering Valve **(EXTERNAL/Boost Link (Valve))**
- Inlet Volume **(EXTERNAL / Boost Link (P))**
- Nozzle Valve **(EXTERNAL / Boost Link (Valve))**

9.2.2.1. Valve Body (linked to Boost)

Valve Body is simulated using Standard Piston element connected via mechanical connection (valve spring) to a Mechanical Boundary element. Gas pressures acting on input and output areas of Piston are received via respective BOOST Link elements (connected to Valve Body with hydraulic connections). Input dialogs of Valve Body and its spring are shown in Figure 9-19.

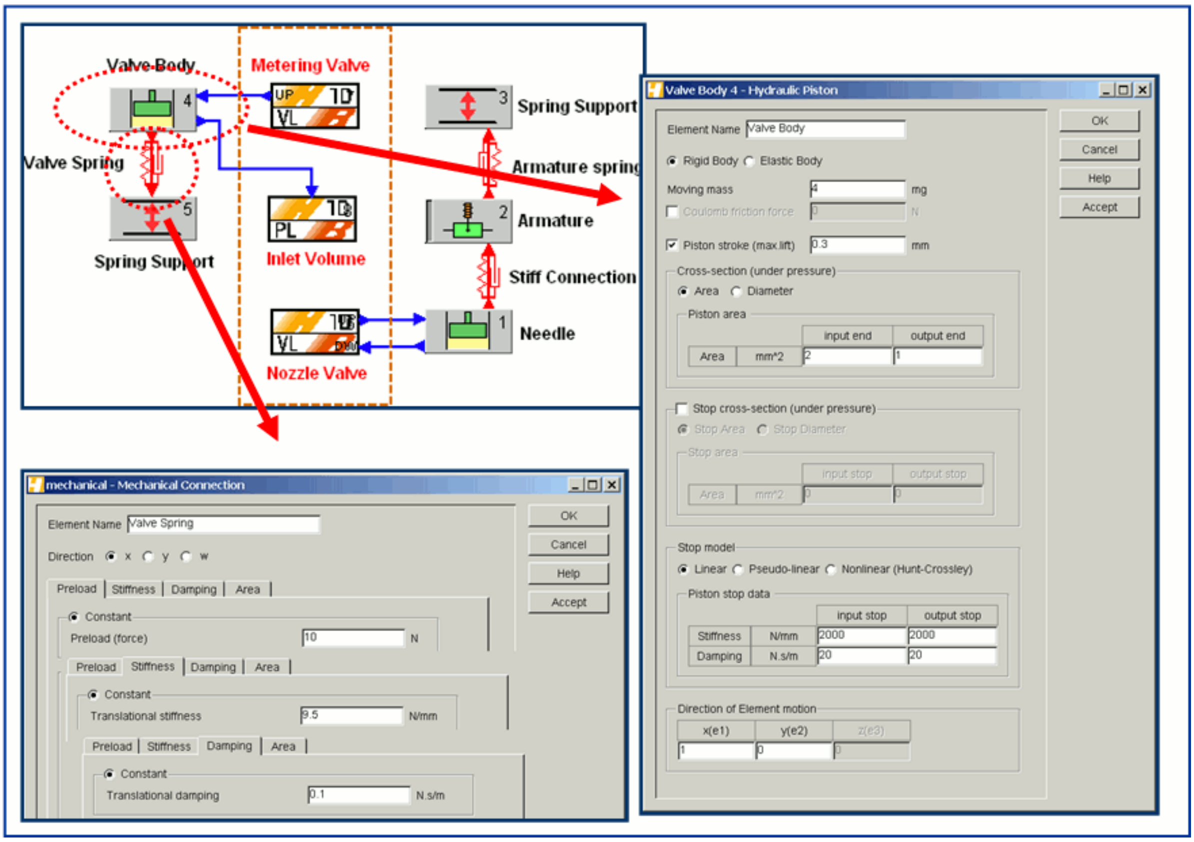


Figure 9-19: Input dialogs of Valve Body and its spring

Valve Body /

Moving mass: 4 mg

Piston stroke: 0.3 mm

Input piston area: 2 mm^2

Output piston area: 1 mm^2

Input stop stiffness: 2000 N/mm

Input stop damping: 20 Ns/m

Output stop stiffness:2000 N/mm

Output stop damping:20 Ns/m

Valve Spring /

Preload: 10 N

Stiffness: 9.5 N/mm

Damping: 0.1 Ns/m

9.2.2.2. Needle (linked to Boost)

Needle is simulated using Standard Piston element rigidly connected to Armature of the solenoid valve. In this way needle lift is directly controlled by the armature motion. Pressurized areas are defined at needle seat position. At this position BOOST calculates gas flow through Nozzle Valve (BOOST Link element) using Needle lift received from BOOST Hydsim.

BOOST Link (Valve) element has two connection anchors: upstream (UP) and downstream (DW). The symbols UP and DW are placed at the connection anchor in the icon of BOOST Link (Valve). In our model, input hydraulic connection of Needle-Piston element is attached to the upstream (UP) anchor and output hydraulic connection - to the downstream (DW) anchor of BOOST Link (Valve). This implies that input end area of Needle-Piston is subjected to upstream gas pressure and output end area of Needle-Piston - to downstream gas pressure. Both pressures are calculated by BOOST.

Needle /

Moving mass: 6 mg

Input end diameter: 2 mm

Output end diameter:2 mm

Input stop stiffness: 2000 N/mm

Input stop damping: 20 Ns/m

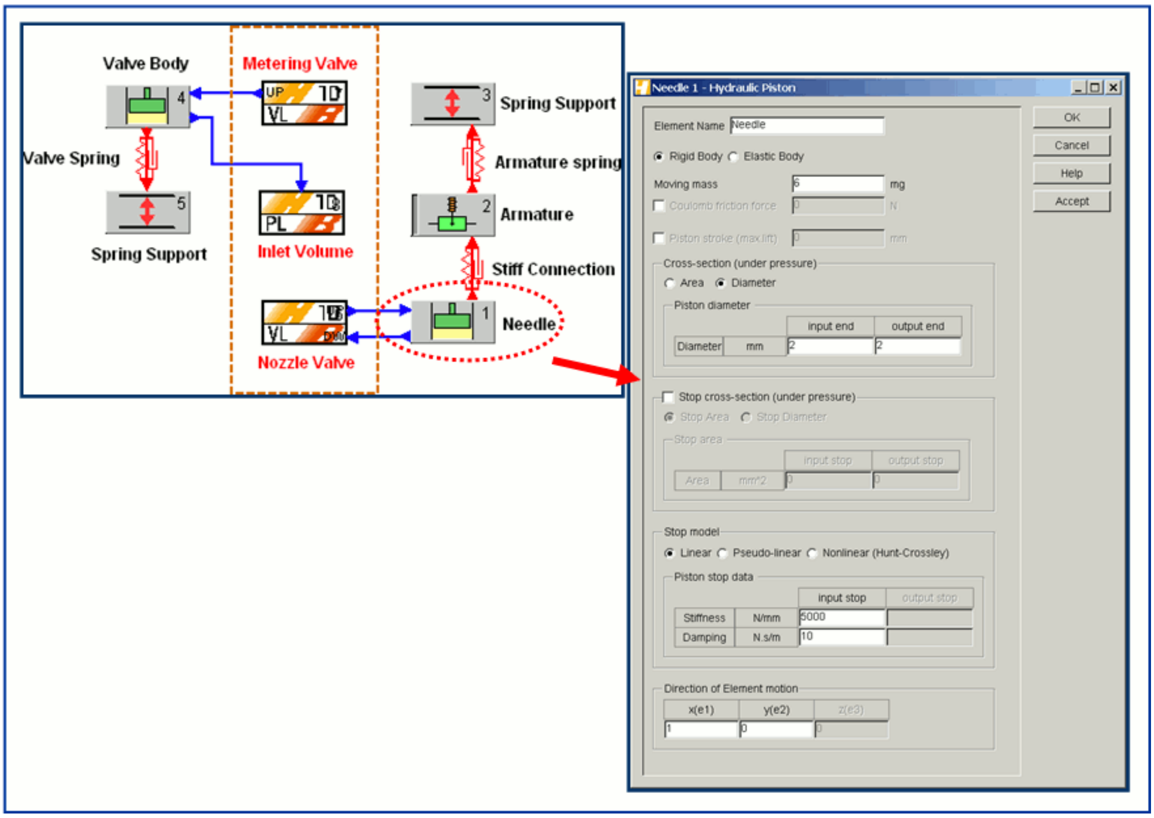


Figure 9-20: Input dialog of Needle-Piston

9.2.2.3. Armature

Solenoid Armature is an electromechanical element driven by a magnetic force. For simplicity, in our model magnetic force is chosen to be constant (boundary condition). Switch ON/OFF events of magnetic force are specified in the solenoid timing table. Force magnitude has to be high enough to open the valve against the armature spring. When solenoid is switched off (magnetic force set to zero), armature spring returns the armature and needle to the seat position and thus closes the nozzle. Input dialogs of Armature and its spring are shown in Figure 9-21.

Armature /

Moving mass: 4 mg

Armature stroke: 0.3 mm

Magnetic force: 70 N

Switch ON event: 0.1 ms

Switch OFF event: 0.8 ms

Body diameter on input end: 3 mm

Body diameter on output side: 3 mm

Input stop stiffness: 50000 N/mm

Input stop damping: 10 Ns/m

Output stop stiffness:50000 N/mm

Output stop damping: 10 Ns/m

Armature Spring /

Preload: 50 N

Stiffness: 50 N/mm

Damping: 0.3 Ns/m

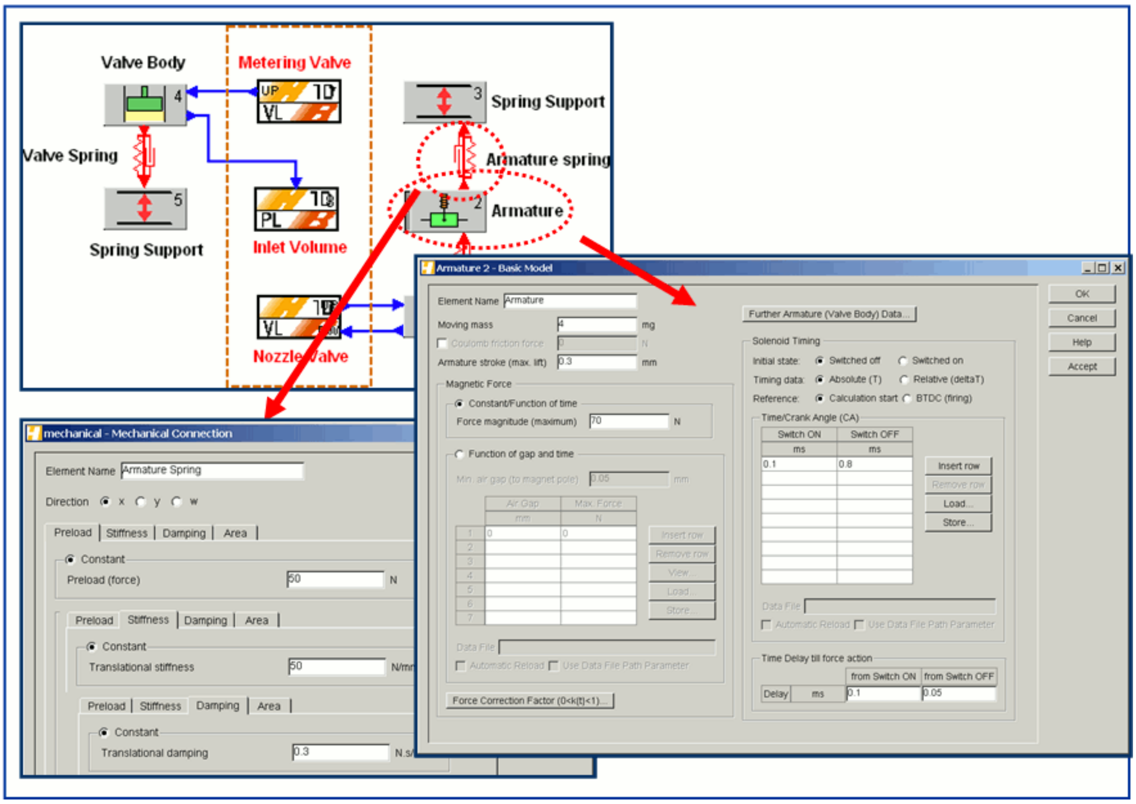


Figure 9-21: Input dialog of Solenoid Armature

9.2.2.4. Metering Valve (Boost Link)

Metering Valve is an interface element of BOOST Link (Valve) type. It is connected by the hydraulic connection to the input end of Valve Body (piston). In this way the input end of Valve Body is subjected to the upstream (UP) pressure from its counterpart element CV1 in the BOOST sub-model. Displacement (x-coordinate) of input end of Valve Body is communicated to BOOST. Communication takes place through 1st channel. Input dialog of Metering valve is shown in Figure 9-22.

Number of channels in the communication between BOOST and BOOST Hydsim is defined by the number of BOOST Link elements in BOOST Hydsim. The user can decide himself which channel to assign to which counterpart element in the BOOST sub-model.

Metering Valve /

Channel Number: 1

Position of valve body: x-coordinate of upper piston input end (sent to BOOST)

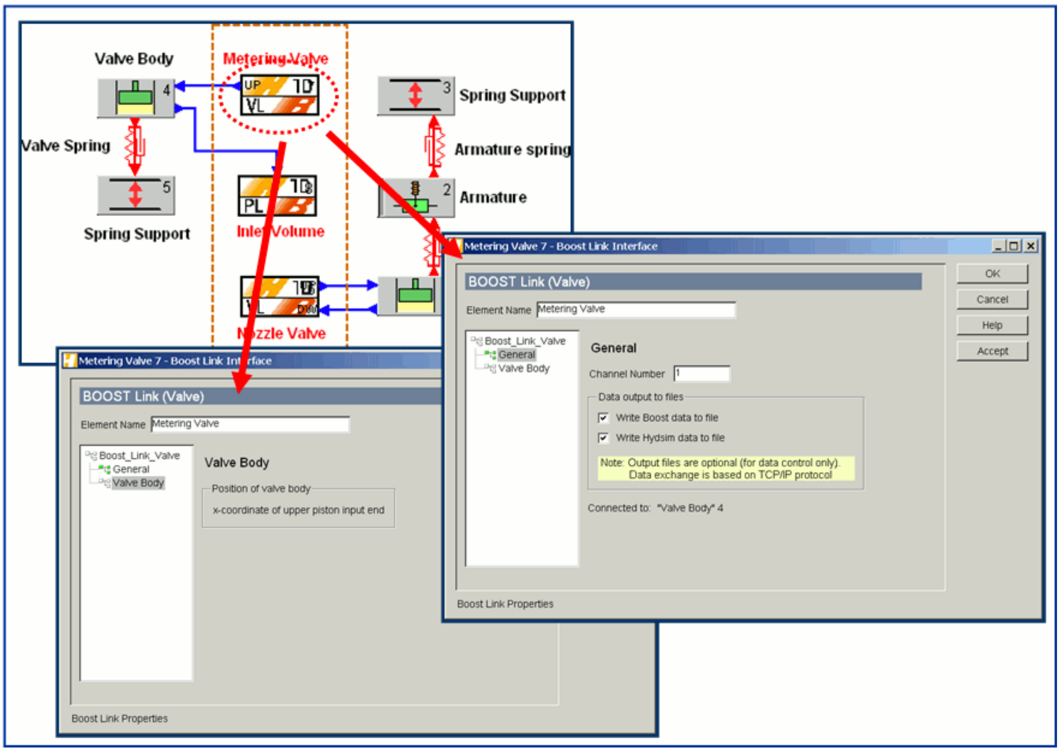


Figure 9-22: Input dialog of Metering Valve (interface element)

9.2.2.5. Inlet Volume (Boost Link)

Inlet Volume is an interface element of BOOST Link (Plenum) type. It is connected by the hydraulic connection to the output end of Valve Body (piston). BOOST Link (Plenum) element can be linked only with Plenum elements in BOOST sub-model. Generally BOOST Link (Plenum) element in BOOST Hydsim has same characteristics like any other volume element. It may have up to 10 hydraulic connections. Each Piston element connected to ST Link (Plenum) will generate displacement flow. BOOST Hydsim will sum up these displacements and send the total sum to BOOST. Gas pressure received from BOOST will be applied (through hydraulic connection) on the respective area of connected Piston element. Input dialog of Inlet Volume is shown in Figure 9-23.

Inlet Volume /

Channel Number: 3

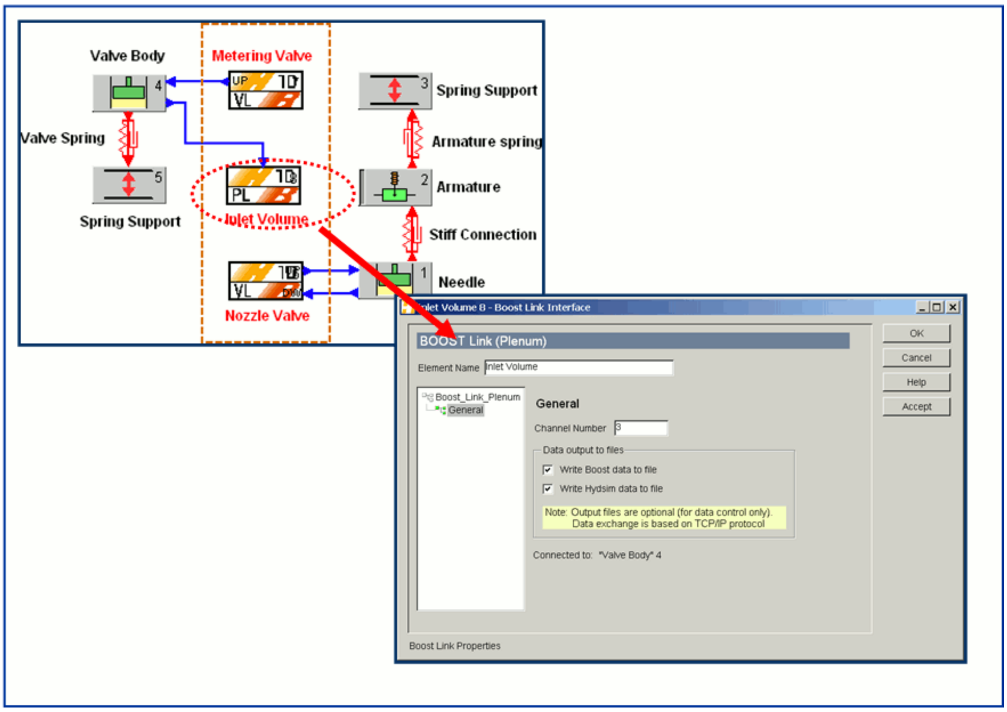


Figure 9-23: Input dialog of Inlet Volume (interface element)

9.2.2.6. Nozzle Valve (Boost Link)

Nozzle Valve /

Channel Number: 2

Position of valve body: x-coordinate of upper piston input end (sent to BOOST)

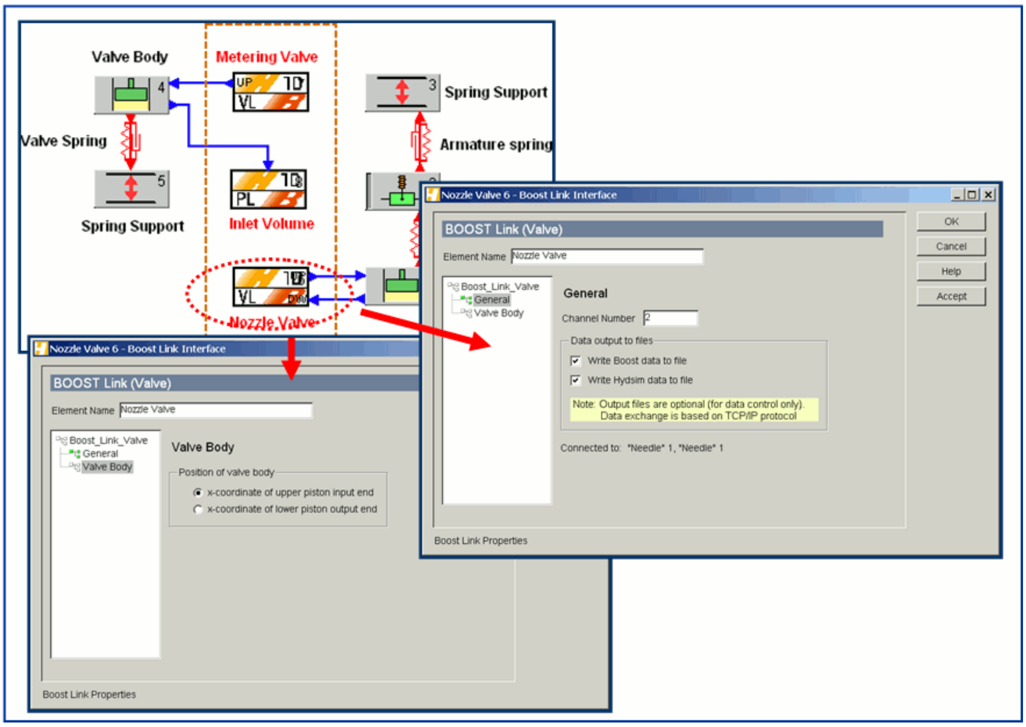


Figure 9-24: Input dialog of Nozzle Valve (interface element)

Nozzle Valve is an interface element of BOOST Link (Valve) type. It is connected by two hydraulic connections to Needle element. Input end of Needle (arrow of the hydraulic connection pointing into Needle icon) is connected to upstream (UP) anchor while output end of Needle (arrow of the hydraulic connection pointing out of Needle icon) is connected to downstream (DW) anchor of Needle Valve. This implies that the input area of Needle is subjected to the upstream (UP) pressure from Nozzle Valve CV2 element in the BOOST sub-model. Analogously, the output area of Needle is subjected to the downstream (DW) pressure from Nozzle Valve CV2. BOOST Hydsim sends to BOOST the coordinate of the Needle input end because this coordinate is activated in the input dialog shown in Figure 9-24. The decision which coordinate should be send to BOOST has to be made by the user. In our case it may be the needle input or output coordinate. However, BOOST can receive only one coordinate of one Piston element. If the Piston is elastic (i.e. it has two different coordinates at input and output end) or if two Piston elements are connected to BOOST Link (Valve), then the user has to choose the appropriate coordinate of one Piston for the transfer to BOOST.

9.2.2.7. Calculation Control

Calculation Control / Simulation /

Reference Speed: 1000 rpm

Engine Speed: 2000 rpm

Output Domain: Crank Angle

Reference angle step: 0.0006 deg

End Reference Angle: 12 deg

Number of values to store: 400

Boost Link / Simulation / ACCI Interface /

Hostname: localhost

Port number: 7777

Exchange step: 0.012 deg

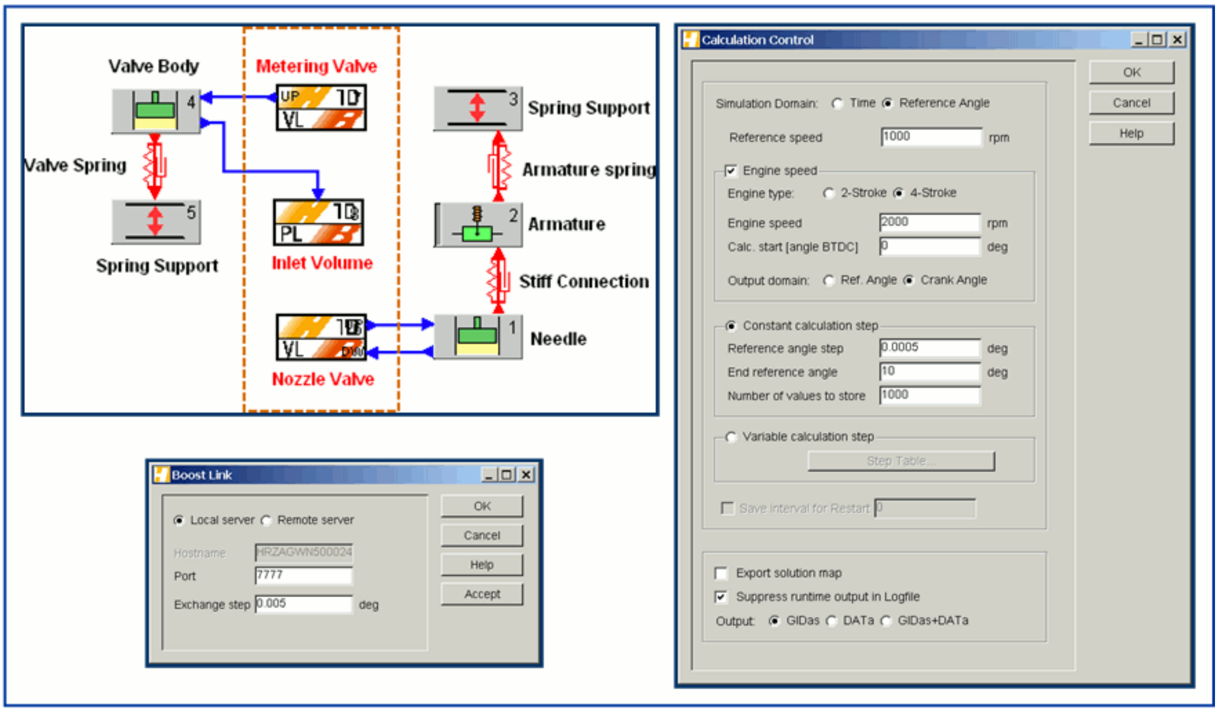


Figure 9-25: Calculation Control and ACCI Interface dialogs

Calculation Control and **ACCI Interface** dialogs are shown in Figure 9-25. There all relevant data necessary for BOOST Hydsim simulation and communication with BOOST are defined. As BOOST uses the crank angle domain for calculation, in BOOST Hydsim the **Crank angle** output domain is selected, too. Note that the data exchange step with BOOST is chosen 10 times larger than the BOOST Hydsim calculation step. To speed up the data exchange via ACCI interface (which is the most time-consuming calculation part), the exchange step can be chosen even up to 20 times larger but the result accuracy has to be controlled. The reason behind it is that gas dynamics is numerically much more "softer" problem than multi-body or fluid dynamics. Start and end of calculation is determined by BOOST Hydsim as co-simulation master.

9.2.3. Co-simulation Results

Selected calculation results in the crank angle domain are shown in Figure 9-26. Note that both BOOST and BOOST Hydsim results are easily postprocessed in the same IMPRESS Chart window.

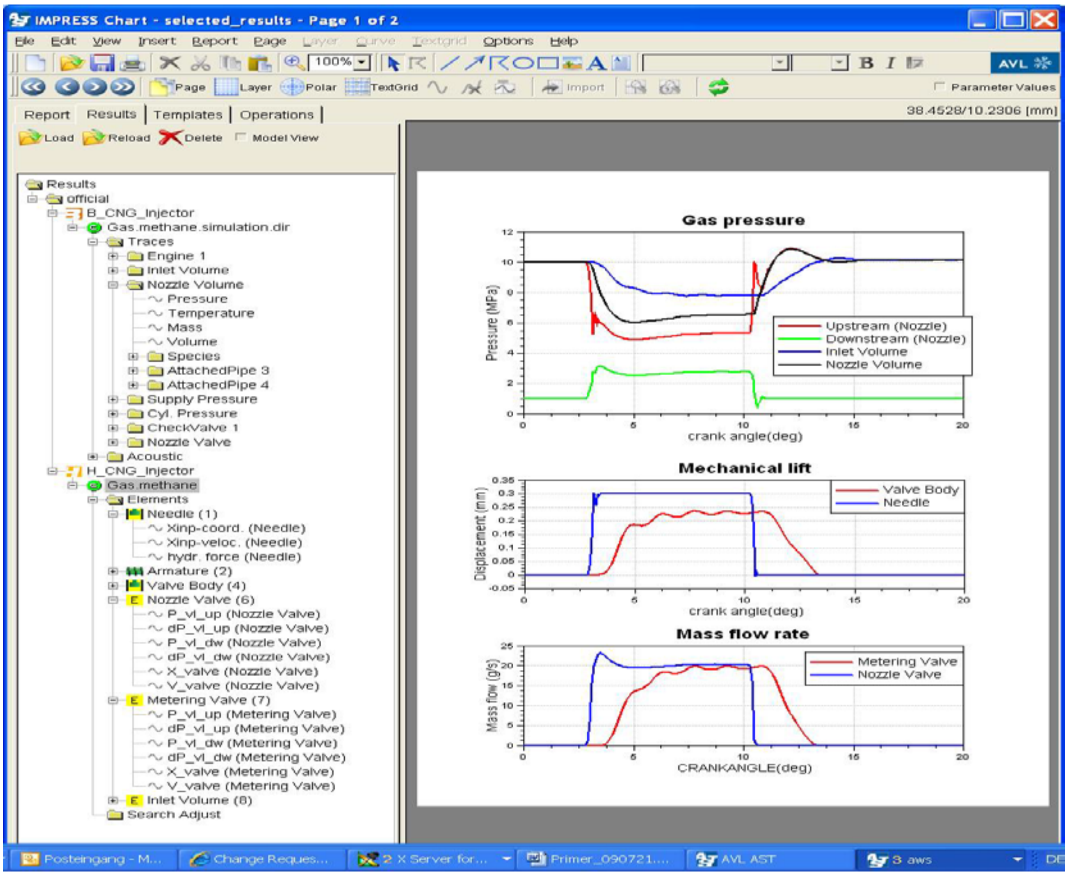


Figure 9-26: Selected results of BOOST Hydsim -BOOST co-simulation

10. BOOST HYDSIM – FIRE LINK

10.1. Introduction

BOOST Hydsim-FIRE Link is an on-line coupling between the BOOST Hydsim injector model and FIRE nozzle flow model. It is designed exclusively for the coupled 1D-3D nozzle flow simulation. For this purpose, there is a **Fire Link (Nozzle)** element in **External** group of BOOST Hydsim element tree. It can be connected via a wire link to two **Nozzle** group elements (only): **Extended SAC Nozzle** and **Extended VCO Nozzle**. Data exchange between BOOST Hydsim and FIRE kernels is carried out via ACCI interface.

The coupled simulation technique works in the following way: BOOST Hydsim controls the co-simulation process and calls the FIRE multiphase solver at the time the needle lift reaches a prescribed tolerance (option 1) or at user-defined data exchange intervals (option 2). Between these intervals, e.g. pilot and main injection, when the needle is closed, the CFD simulation is carried out with a much larger or even one single time step. In this way the computational effort for the simulation of entire injection cycle is minimized. Concerning the physical data exchange, BOOST Hydsim provides to FIRE the current lift of the needle tip and the fuel pressure and temperature at the interface. Based on these data, FIRE moves the computational mesh, adjusts the boundary conditions, computes a time step and sends back to BOOST Hydsim the hydraulic force acting on the needle tip, mass flow rate through the needle seat and spray holes and spatially averaged pressure and temperature in the nozzle sac.

10.2. Common Rail Injector Model with FIRE Link

This chapter guides the user through the BOOST Hydsim-FIRE Link example based on the diesel common rail injector model with Sac nozzle.

10.2.1. BOOST Hydsim Model of Injector

The injector in question is a classical solenoid-controlled injector with 2/2 way valve. Schematic of the injector is shown in Figure 10-1. The injector consists of the rail inlet, high and low pressure bores and volumes, sac-type nozzle with needle, control piston moving in the control chamber, inlet and outlet orifices and control valve actuated by the solenoid armature. Hydraulic boundary conditions for the injector model are the high pressure connection to rail, backflow, leakage pressure and cylinder pressure. Electrical boundary condition is the actuation current of the solenoid coil.

FIRE Link element in BOOST Hydsim is a symbolic icon connected to the Nozzle element by the so-called wire connection. Within its dialog the exact interface position, needle lift tolerance, shaft diameter, time delay till the co-simulation start, exchange time step and other control parameters are defined. FIRE and BOOST Hydsim calculation domains with exchange variables are shown in Figure 10-2. On each exchange step FIRE transfers to BOOST Hydsim the following data:

- F_{Nx} – x-component of needle tip force vector
- q_{seat} – flow rate through needle seat
- q_{holes} - flow rate through spray holes (outlet)

- p_{topV} – boundary pressure below interface line
- T_{topV} – boundary temperature below interface line
- p_{sac} – average pressure in sac volume
- T_{sac} – average temperature in sac volume
- p_{gas} – boundary pressure in spray chamber
- T_{gas} – boundary temperature in spray chamber

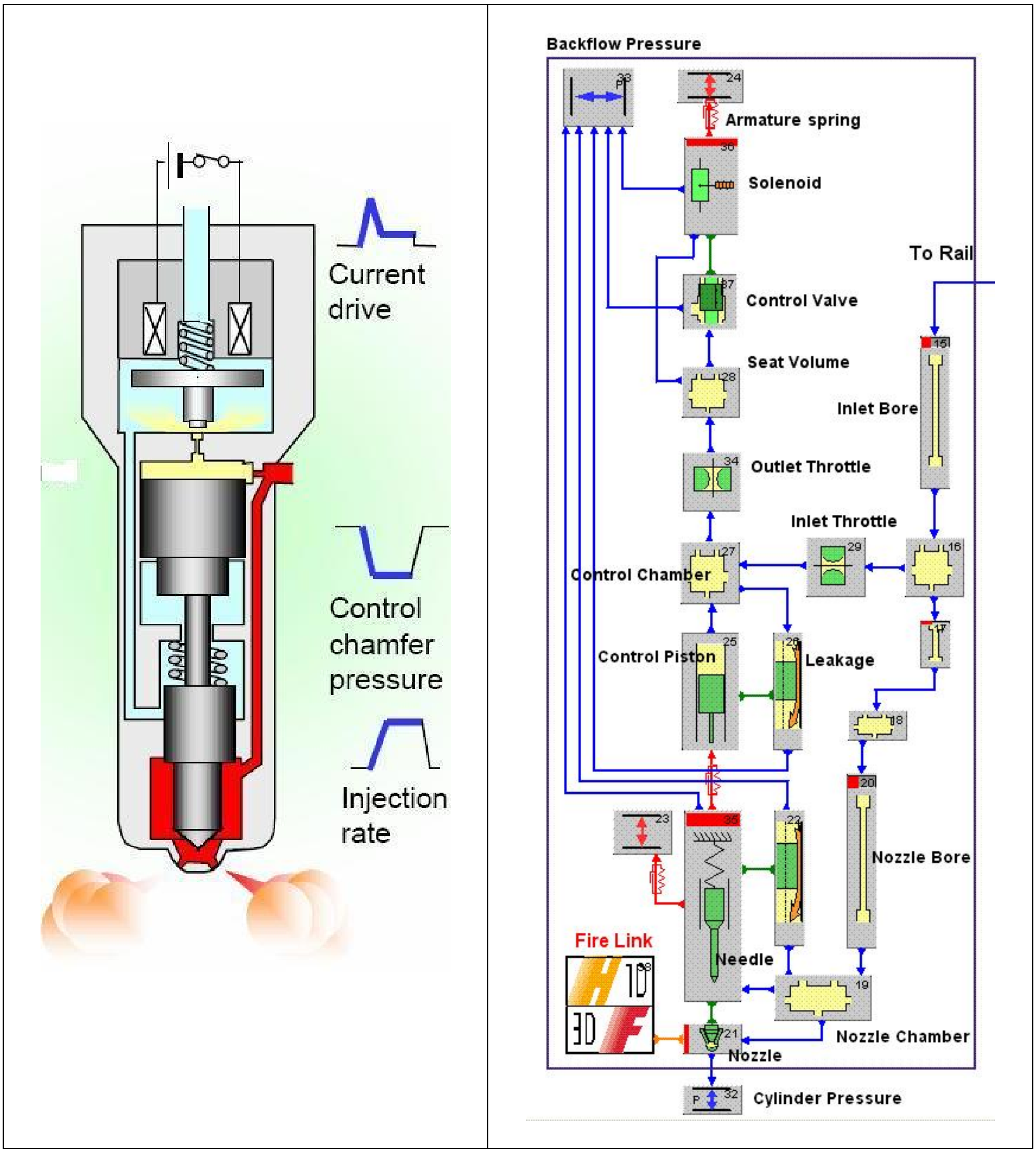


Figure 10-1: Schematic and BOOST Hydsim model of Common Rail injector

In return at the end of the exchange time step BOOST Hydsim transfers to FIRE the following data:

- X_{dx} – x-component of needle tip displacement vector
- p_{nozV} – boundary pressure above interface line
- T_{nozV} – boundary temperature above interface line

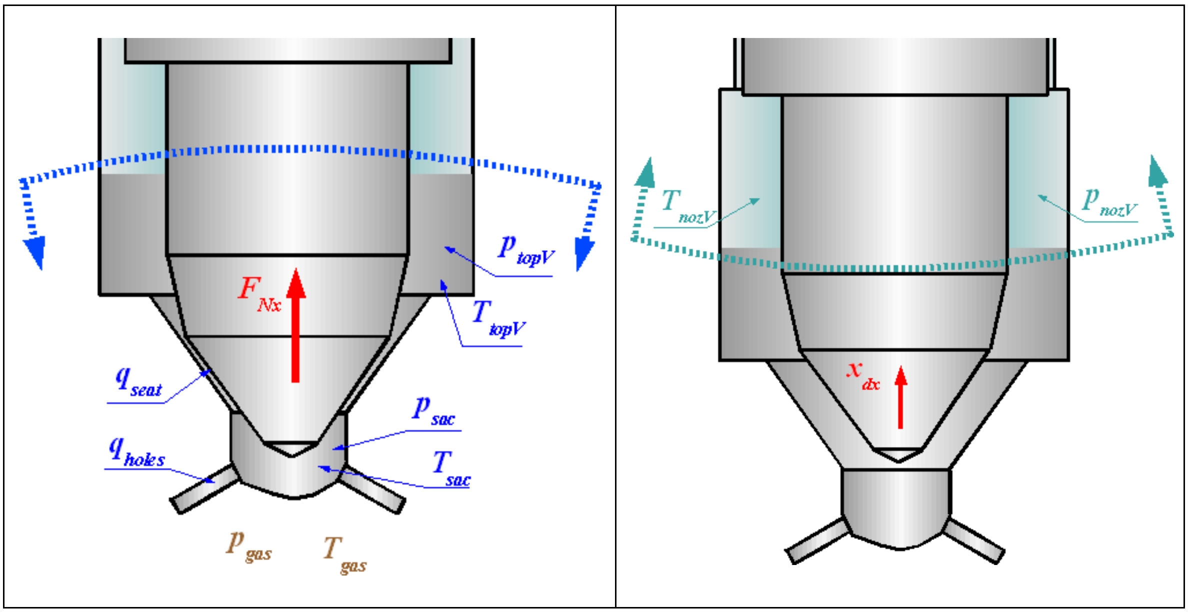


Figure 10-2: FIRE and BOOST Hydsim Calculation domains with exchange variables

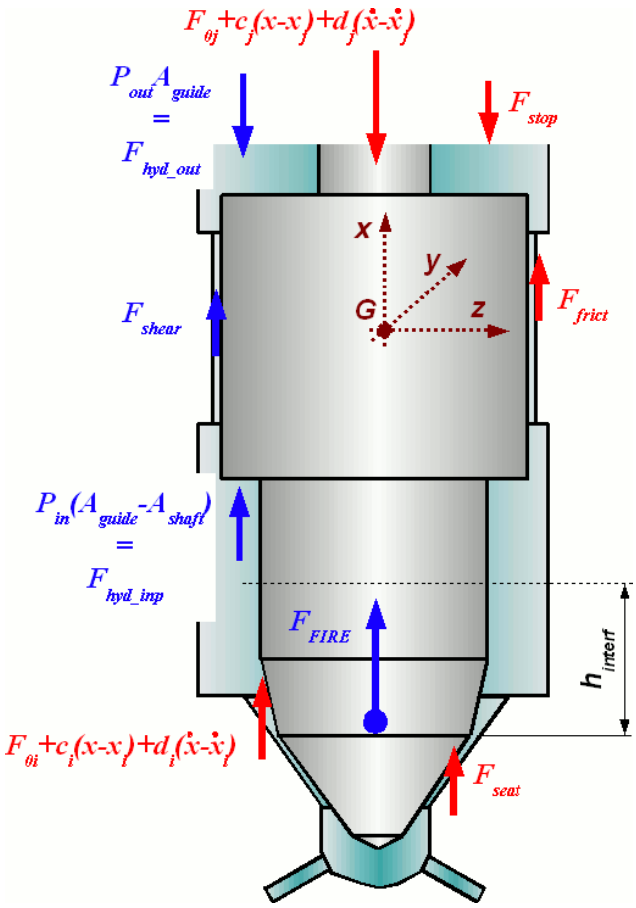


Figure 10-3: Forces acting on Needle

In the actual version of the coupling only the longitudinal x-component of the needle tip force and displacement vectors are used. Needle lift tolerance for the FIRE solver call is defined in this example by 4 µm. BOOST Hydsim calculation step is set to 10-7 s, exchange step with FIRE is 20 time larger (2x10-6 s).

This step is applied only for the intervals where needle lift exceeds the tolerance. Outside of these intervals (i.e. practically between the injection events) 5 times smaller or even one single time step is used.

Schematic of injector needle with hydraulic (blue color) and mechanical forces (red color) acting in longitudinal direction is shown in Figure 10-3. These forces can be classified as follows:

- F_{FIRE} – force on needle tip imported from FIRE
- F_{hyd_inp} – input pressure force acting on needle guide from nozzle volume
- F_{shear} – viscous shear force at guide due to leakage
- F_{hyd_out} – output pressure force on needle back side from back volume
- F_{seat} – needle reaction force at contact with seat
- F_{stop} – needle reaction force at contact with stop
- F_{frict} – Coulomb friction force at needle guide
- $F_0+cx+dx'$ - mechanical connection force from the neighboring element

10.2.2. FIRE Nozzle mesh

Figure 10-4 shows the computational grid used in the CFD simulation. The whole domain consists of the hollow cylinder around the needle shaft, needle seat area with nozzle sac and six holes. To reduce the computational effort, only one of twelve periodic 3D-degree-segments is modeled. The cake-mesh of the nozzle sac and hole is shown in Figure 10-5.

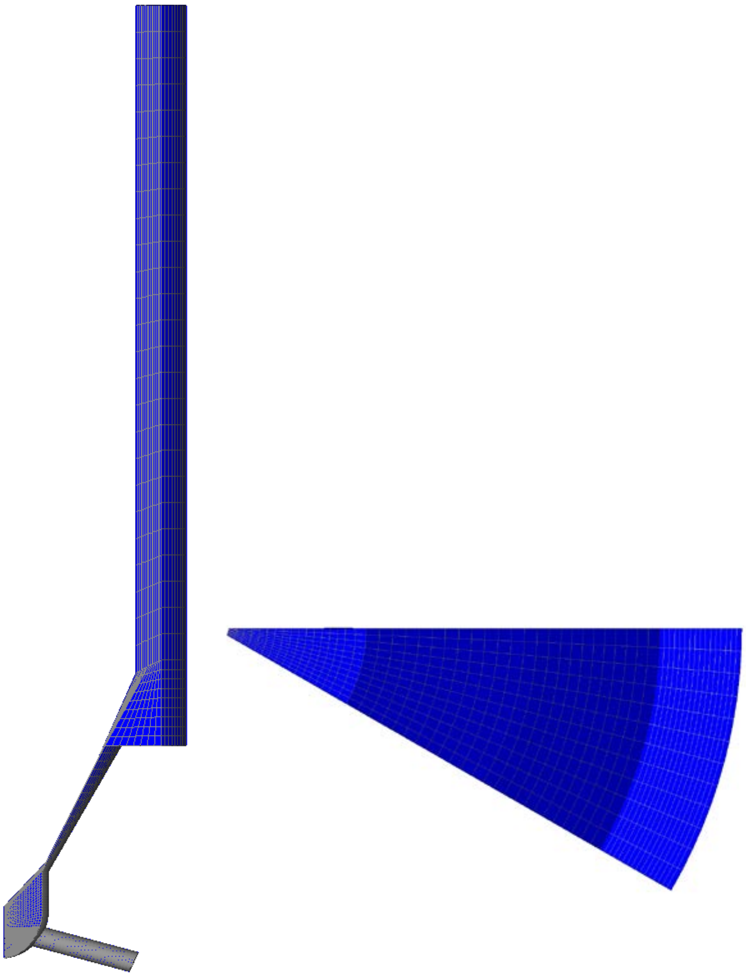


Figure 10-4: SAC-nozzle cake-mesh, front view (left) and top view (right)

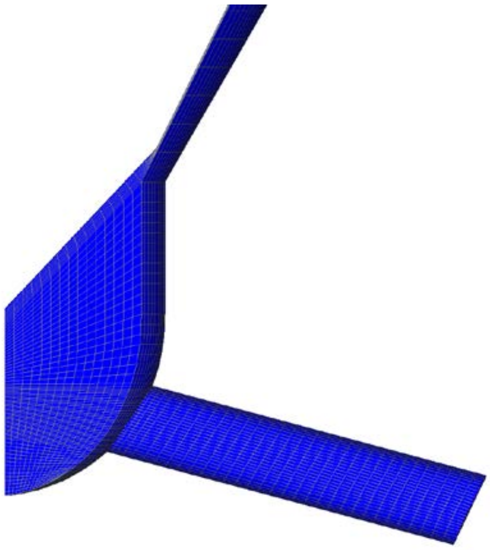


Figure 10-5: SAC-nozzle cake-mesh, nozzle sac region with nozzle hole

The mesh consists of 33620 hexahedral cells with 12 cell layers in the circumferential direction so that one layer corresponds to 2.5 degrees. The outlet bore is meshed separately and connected to the sac-volume by arbitrary-interfaces. The nozzle gap itself contains 31 cell rows in the radial direction (which is the main flow direction) and 8 rows in vertical direction (normal to the main flow direction). These cells are from 10 (at the seat edge) to 90 μm long and 12.5 μm high at needle lift of 100 μm. This mesh topology is chosen according to the AVL experience with the 3D nozzle flow simulation.

The needle movement is accomplished by the mesh-deformation at solver run-time. Only one mesh is created in a reference position. The mesh-deformation-function shifts the needle surface at each time step according to the needle displacement received from BOOST Hydsim. After that, the position of the internal nodes is updated by the Laplace interpolation scheme. To prevent collapsing cells at zero or very small needle lift, a minimal gap size (5 μm in the actual case, measured normal to the needle seat) is maintained in the mesh. The needle movement below this minimal gap is simulated by assigning a very high flow resistance to all those cells that are actually located within the solid needle.

For more details of FIRE nozzle mesh please consult Verification Report – FIRE-BOOST Hydsim Online Coupling, SAC Nozzles.

10.2.3. FIRE Nozzle Interface Data

FIRE Nozzle interface is represented by a symbolic **FIRE Link** element in BOOST Hydsim. The co-simulation data have to be defined in its **Properties** dialog. This dialog consists of **General** and **Control** sheets shown in Figure 10-6 and Figure 10-7, respectively. In **General** sheet and the user has to define the interface border height from needle seat (any position between seat and guide can be chosen), needle shaft diameter at interface and (optionally) names of data output files with FIRE and BOOST Hydsim exchange data (very useful for cross-checking). In **Control** sheet the exchange time/angle step (at open needle) between BOOST Hydsim and FIRE has to be specified. Usually, it should be 10 to 20 times higher than BOOST Hydsim calculation step.

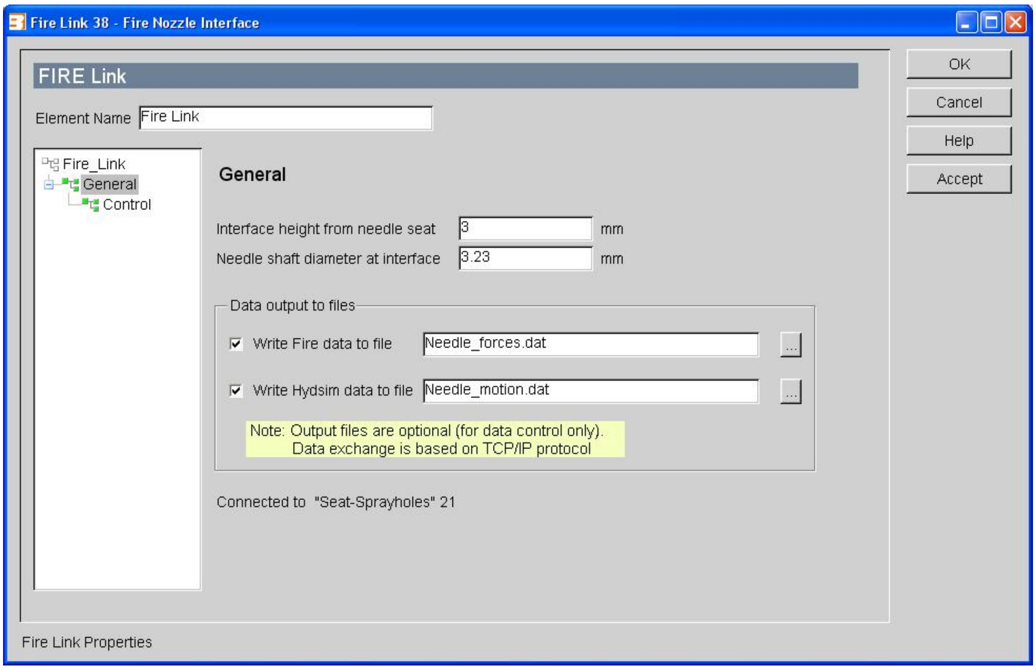


Figure 10-6: FIRE Link General Dialog

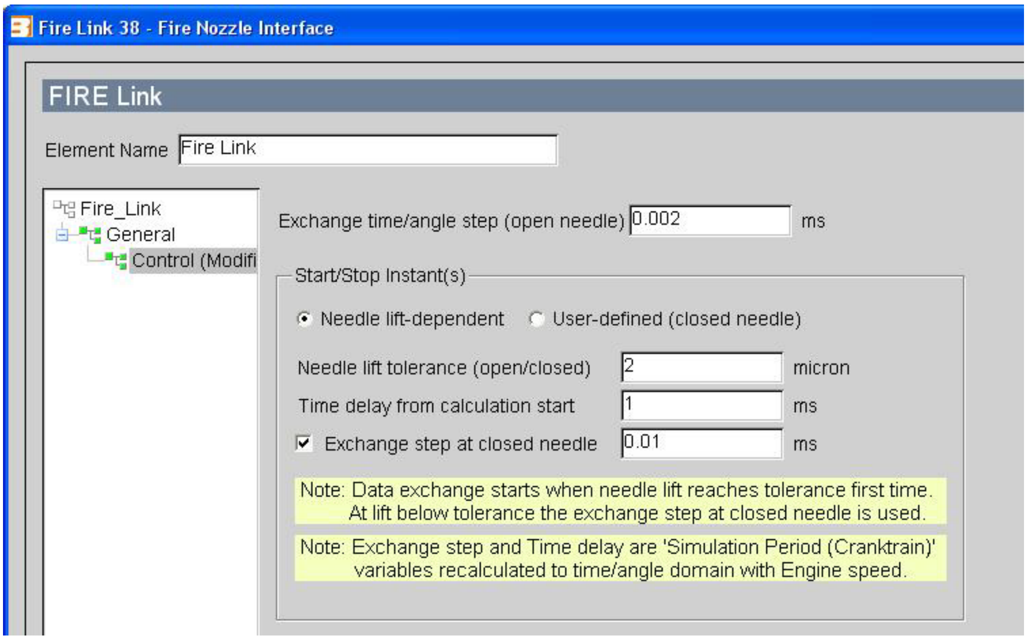


Figure 10-7: FIRE Link Control Dialog

Within **Control** sheet two options of co-simulation with FIRE solver can be chosen. First, **Needle-lift dependent**, option implies initial call of the FIRE solver (start of data exchange) at user-defined needle tolerance (2 µm in our example). At needle lift below tolerance the exchange step at closed needle (optional parameter) is used. If not defined, it is set by the program to the exchange step at open needle. Additionally parameter **Time delay from calculation start** has to be specified (1 ms in our example). BOOST Hydsim-FIRE co-simulation cannot start earlier than this time interval expires. This parameter is required to prevent the unexpected co-simulation begin at start where BOOST Hydsim model may not be in the equilibrium position (i.e. needle may have initial lift due to wrong initials conditions). Second, **User-defined (closed needle)**, option simply implies calling of the FIRE solver at user-defined time/angle intervals.

Between the intervals a single (i.e. usually very high) exchange time step is used. In this way, the CFD calculation is skipped from the end of the previous interval till the beginning of the next interval. This allows speeding up the simulation procedure considerably. However, the user has to take care that the specified intervals fully cover the injection events. Otherwise, if e.g. the needle is already open at the interval start or not yet closed at the interval end, an error message will be produced.

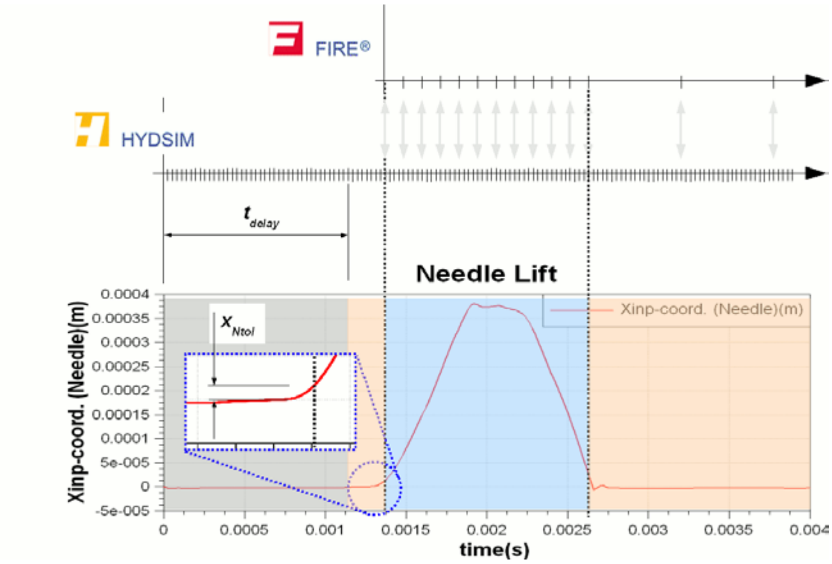


Figure 10-8: Co-simulation with Needle-lift dependent option

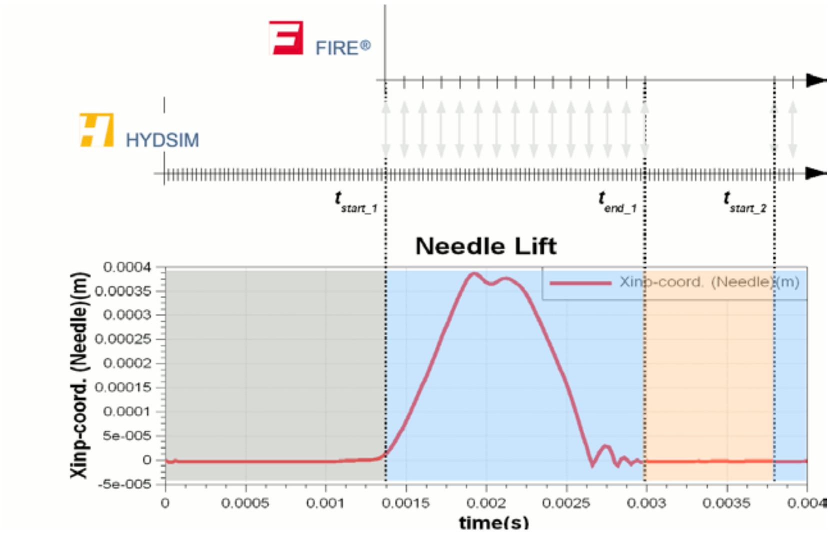


Figure 10-9: Co-simulation within User-defined time interval(s)

10.2.4. Running Co-simulation

Before starting the BOOST Hydsim- FIRE co-simulation, the name and port number of the ACCI server have to be specified within the ACCI Interface/FIRE Nozzle dialogs shown in Figure 10-10.

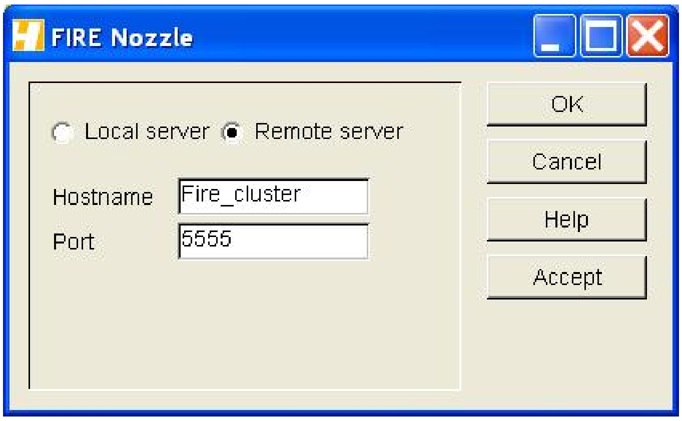


Figure 10-10: ACCI interface FIRE Nozzle dialog

The same server name and port number have to be defined in the FIRE case, too. Then standard BOOST Hydsim simulation with Run command can be performed. Restart calculation is also supported.

10.2.5. Coupled Simulation Results

BOOST Hydsim calculates the injector flow till interface border and mechanical motion of the needle, control piston and solenoid armature. For this, hydraulic, mechanical, electromagnetic and shear forces are calculated first within the entire system. The flow rate through needle seat and spray holes is obtained from FIRE. Depending on the flow regime (laminar or turbulent) this flow rate is compared with the simplified 1D flow theory. Basically, discharge coefficient at the narrowest cross-sectional area of the needle seat is estimated and checked for limiting range. If for the turbulent flow regime this coefficient exceeds maximal value (1), a warning message is produced in **View Logfile** window (actually the 5% error is formally allowed).

Within the actual example, two injection events (pilot and main) are modeled. Pressures in the nozzle and control chambers and seat volume are depicted in Figure 10-11. Lift of the solenoid armature, control piston and needle is shown in Figure 10-12. One can observe that the delay between the solenoid actuation (armature motion) and needle lift start is 0.2 ms. At pilot injection the needle lift is always ballistic. The control piston is modeled as an elastic body, therefore its initial lift is negative because the piston is compressed by the rail pressure in the control chamber. As soon as the control valve opens, the pressure in the control chamber is gradually released. This causes the elastic relaxation of the control piston and the delayed opening of the needle tip. Hydraulic forces acting on needle tip (from FIRE) and guide (from BOOST Hydsim) are depicted in Figure 10-13. Flow through the inlet and outlet orifices and nozzle (injection rate) is plotted in Figure 10-14. Note that by performing BOOST Hydsim calculation standalone, flow discharge coefficients at needle seat and spray holes have to be specified on input. These coefficients are unknown and usually can be obtained only by the specific experiment. With BOOST Hydsim-FIRE co-simulation, these coefficients are not required: they are estimated automatically as the by-product of the calculation. This is a big advantage of the coupled 1D-3D simulation over the standard 1D simulation alone.

Valuable information on the nozzle cavitation can be extracted from the effective cross-sectional flow areas at needle seat and nozzle holes. These flow areas (geometric and effective) are depicted in Figure 10-15. The total geometric area of six nozzle holes is 0.24 mm^2 (constant value). Total effective area of holes varies from 0.161 mm^2 at turbulent non-cavitating flow to 0.146 mm^2 at cavitating flow. Effective flow area at needle seat is a function of nozzle geometry and needle lift. It reaches the effective holes area of 0.146 mm^2 at 90 μm needle lift for both pilot and main injections. At this lift the cavitation transition from the needle seat to nozzle holes gradually occurs. Note that FIRE time counter is shifted from the BOOST Hydsim time by 1.14 ms because FIRE calculation is started at needle lift of 5 μm (user-defined tolerance). In other words, BOOST Hydsim time of 1.14 ms implies zero time in FIRE.

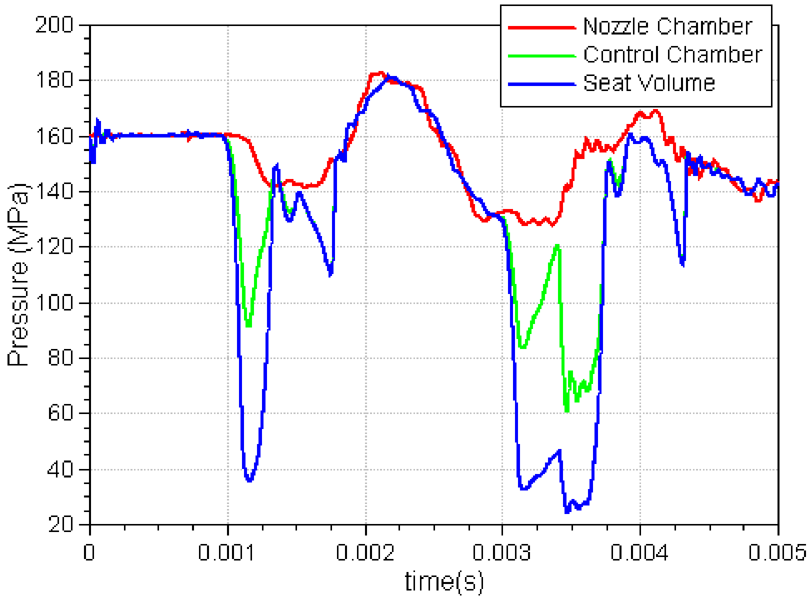


Figure 10-11: Pressure in nozzle and control chambers and seat volume

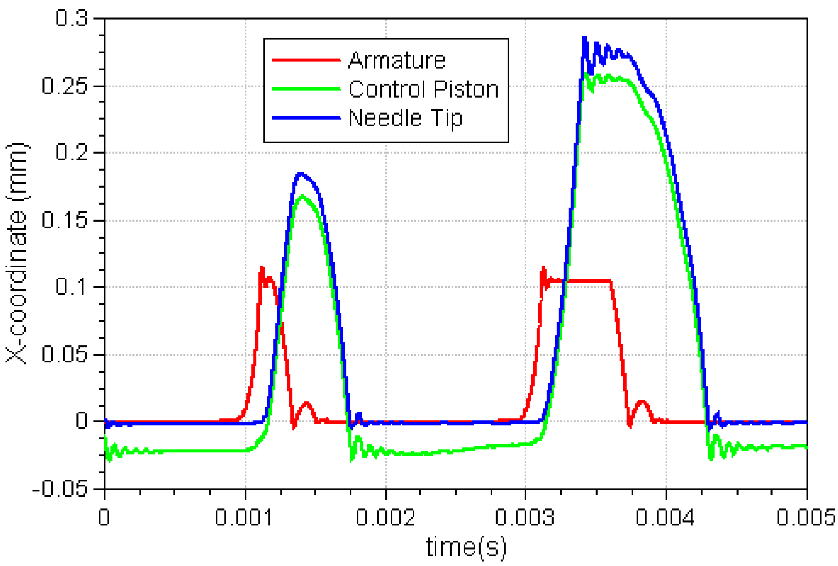


Figure 10-12: Motion of solenoid armature, control piston and needle tip

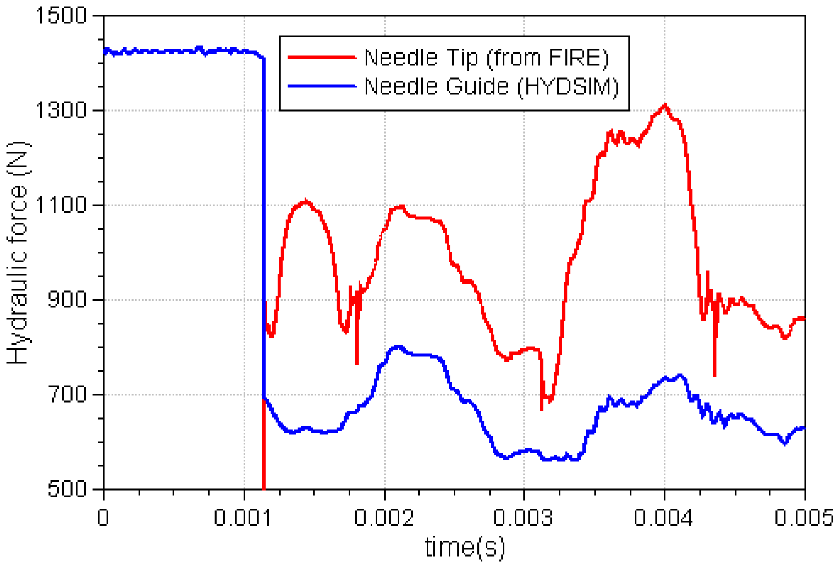


Figure 10-13: Hydraulic force on needle tip (FIRE) and guide (BOOST Hydsim)

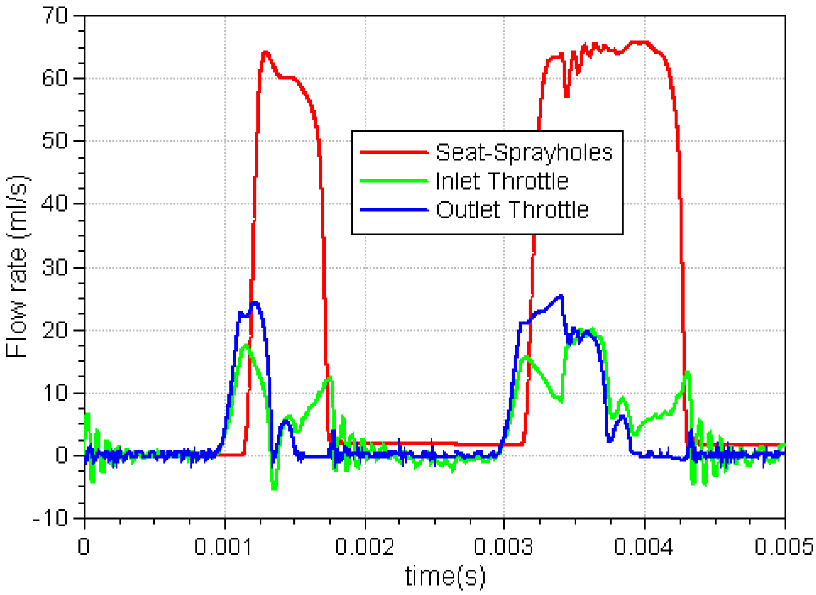


Figure 10-14: Flow rate through nozzle (seat and holes) and inlet/outlet throttles

From the FIRE calculation we can derive the average flow discharge coefficient values at needle seat and spray holes. For this the classical Bernoulli equation is used, both for cavitating and non-cavitating turbulent flow regime. In this way, the needle seat discharge coefficient value is estimated as 0.9. Nozzle hole discharge coefficients are identified to be 0.67 (at non-cavitating turbulent flow) and 0.61 (at cavitating flow), respectively. Using the identified values, the BOOST Hydsim standalone calculation with the 1D nozzle flow model is performed and good correlation with the results of 1D-3D coupled simulation is attained. Needle lift in both pilot and main injection is almost identical for both 1D and 1D-3D simulation cases. Injection rates also fit together reasonably well as shown in Figure 10-16. This leads to the important conclusion that the flow discharge coefficients, derived from coupled 1D-3D simulation, can be successfully used for the further 1D nozzle calculation with BOOST Hydsim alone.

Obviously this implies a substantial saving of the computational time. For comparison, coupled 1D-3D simulation of the actual injector with the optimized calling procedure of the CFD solver requires 7 to 8 hours CPU time on a typical PC Intel Core 2 while 1D simulation with BOOST Hydsim alone takes only 15 to 30 s.

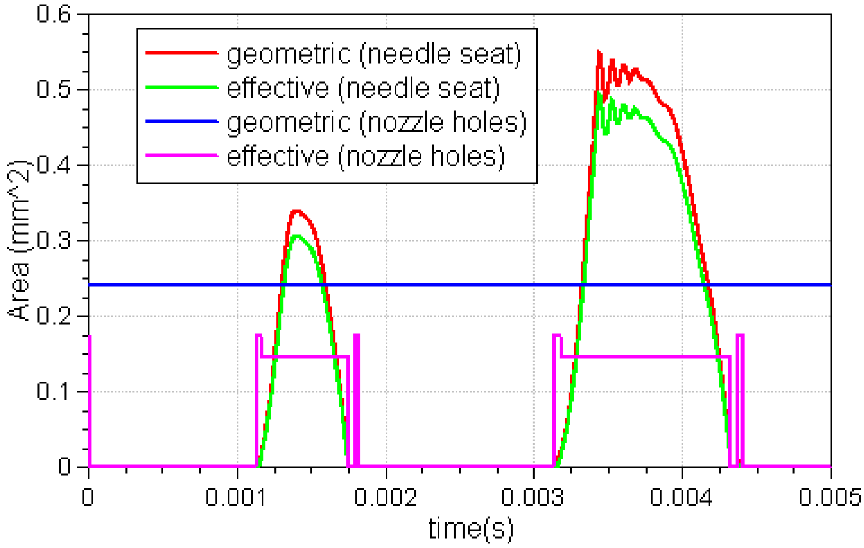


Figure 10-15: Geometric and effective flow area of needle seat and nozzle holes

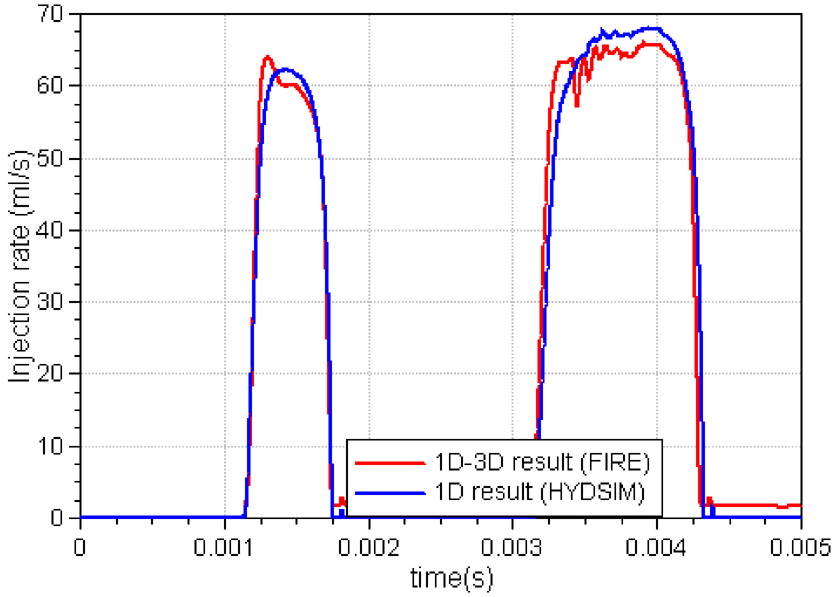


Figure 10-16: Injection rate from 1D-3D co-simulation and 1D simulation alone with adjusted flow discharge coefficients

11. CONCLUDING REMARKS

This Primer is intended as an extended introduction for a BOOST Hydsim user. It describes the modeling and analysis of different types of fuel injection and hydraulic valve train systems, which are commonly used in automotive, marine propulsion, and other combustion engines. However, the Primer does not include the advanced models of complete injection systems, e.g. multiple injectors, complete high-pressure fuel supply systems, peripheral devices, etc. These systems, of course, can also be modeled with BOOST Hydsim, but may require certain prior experience and substantial computational effort. Their handling is explained in the BOOST Hydsim training courses.

The capabilities of BOOST Hydsim are not limited to modeling of fuel injection systems and hydraulic valve trains. In principle, BOOST Hydsim can be applied to the dynamic analysis of any type of hydraulic and hydromechanical system, as long as one-dimensional flow modeling is sufficient. Such types of systems are e.g. fuel storage, supply or filling/spill systems, hydraulic power transmission and control units etc. This Primer does not cover the modeling of hydraulic or hydro-mechanical systems other than fuel injection systems and hydraulic valve trains.

Till recent time BOOST Hydsim was usually used as stand-alone software for the simulation of hydraulic systems. However, now it became an even more powerful tool for engine analysis when combined with other AVL products: FIRE Multi-phase Flow and Spray modules, BOOST for engine cycle simulation and EXCITE for engine vibration and acoustic analysis. Recently the coupling interfaces between BOOST Hydsim and FIRE for the nozzle flow simulation and BOOST Hydsim and BOOST for the hydromechanical-gaseous system simulation have been developed. Data-exchange interfaces to other AVL products can be developed upon the user request.

The delivery of BOOST Hydsim software contains all examples described in this Primer: simple hydraulic system, common-rail injectors with two-way and three-way solenoid valves, unit injector, mechanical and electronically-controlled in-line pump systems, electronic unit pump, electro-hydraulic valve trains and common rail models with PID controllers. These and many other examples are located in the directory <install_dir>/examples/BOOST Hydsim. They can be directly loaded and executed in AVL Workspace/BOOST Hydsim as soon as it is installed.

The list of main examples delivered with the BOOST Hydsim installation is provided in the Appendix.

12. APPENDIX

Below you can find a list of typical examples delivered with the BOOST Hydsim installation. Examples are located in the directory <install_dir>/examples/BOOST Hydsim/vxxxx/<examples group>. The list of BOOST Hydsim examples is being constantly extended. Therefore a number of similar additional examples (not included into the below table) can be found in the installation.

BOOST HYDSIM Examples

825_Aftertreatment

Urea_dosingsystem_1.hyd			
Urea_dosingsystem_2.hyd			

870_BOOST Link

B_CNG_Injector.bwf	Primer	Boost file	
B_Piston_Plenum.bwf		Boost file	
B_Valve_Piston.bwf		Boost file	
H_CNG_Injector.hyd	Primer	Hydsim file	ACCI link
H_Piston_Plenum.hyd		Hydsim file	ACCI link
H_Valve_Piston.hyd		Hydsim file	ACCI link

845_Control Valve

Check_valve1_std.hyd			
Check_valve2_std.hyd			
Check_valve.hyd			
Control_valve1.hyd			
Control_valve2.hyd			
Spring_valve_std.hyd			

835_Engine brake

engine_brake_6cyl.hyd			
engine_brake_valve_act.hyd			
engine_brake_valve_gas.hyd			

801_Fuel Injection – diesel

Comrail_piezo1_mrc.hyd	Primer		
Comrail_piezo1_mrc_therm.hyd			
Comrail_piezo1_svd.hyd	Primer		
Comrail_piezo2_mrc.hyd			
Comrail_piezo2_svd.hyd			
CR13_piezo1_Btube_ap.hyd			
CR13_piezo2_Btube.hyd	Primer		
CR13_piezo2_mctip.hyd			
HP_pump_shaft_2.hyd			
HP_pump_shaft_3.hyd			
common_rail2wv.hyd	Primer		
common_rail2wv_ap.hyd			
common_rail2wv_ap_2.hyd			
common_rail2wv_therm.hyd			
common_rail3wv.hyd	Primer		
common_rail3wv_2.hyd			

common_rail3wv_ap.hyd			
common_rail32sw.hyd			
comrail2wv_3cases.hyd			
CRail2wv_spray_nozzle.hyd			
CRail2wv_spray_orifice.hyd			
CRail_Injector1.hyd			
CRail_Injector1_therm.hyd			
CRail_Injector2.hyd			
CRail_Injector2_fdb.hyd			
CRail_Injector3.hyd			
CRail_Injector3_ap.hyd			
CRI_Btube_Fm_ext.hyd			
CRI_Btube_Fm_int.hyd			
CRI_RSN_leakages.hyd			
CRIP2_injector_Fm_el_int.hyd			
CRIP2_injector_Fm_el_int_pd.hyd			
CRIP2_injector_Fm_el_int_th.hyd			
CRIP2_injector_Fm_int.hyd			
CRIP2_injector_Fm_int_pd.hyd			
RSN_SAC_nozzle.hyd			
RSN_VCO_nozzle.hyd			
CR_system4_bends.hyd			
CR_system6_bends.hyd			
CR_system_V12.hyd			
CR13_piezo2_4inj.hyd			
CR13_piezo3_4inj.hyd			
distributor_pump.hyd			
distributor_pump_obsn.hyd			
distributor_pump_therm.hyd			
radial_pump.hyd			
radial_pump_inv_con.hyd			
radial_pump_obsn.hyd			
radial_pump_therm.hyd			
VE_pump_fire.hyd			
VP44_pump.hyd			
hpi_tp_injector1.hyd			
hpi_tp_injector1_therm.hyd			
hpi_tp_injector2.hyd			
hpi_tp_injector2_therm.hyd			
cam_contact_loss.hyd			

electronic_pump.hyd	Primer		
electronic_pump_cases.hyd			
electronic_pump_inv_con.hyd			
electronic_pump_nozzle.hyd			
Inline_pump1.hyd	Primer		
Inline_pump1_ap.hyd			
Inline_pump2.hyd			
Inline_pump.hyd			
Inline_pump_ap.hyd			
PLD_GRV.hyd			
plunger_basic_inline.hyd			
plunger_head_groove.hyd			
plunger_relief_grinding.hyd			
pde_awk_4cases_1.hyd			
pde_awk_4cases_1fdb.hyd			
pde_awk_4cases_2.hyd			
pde_awk_4cases_2fdb.hyd			
timing_unit_inj_4cyl.hyd			
unit_inj_shaft_rocker.hyd			
unit_inj_sid_piston1.hyd			
unit_inj_sid_piston2.hyd			
unit_injector.hyd	Primer		
unit_injector_E31.hyd			
unit_injector_obsn.hyd			
unit_injector_restart.hyd	Primer		
unit_injector_rocker1.hyd			
unit_injector_rocker.hyd			
unit_injector_shaft.hyd			
unit_injectors_4.hyd			
805_Fuel Injection – dimethyl_ether			
common_rail_4dme.hyd			
common_rail_4dme_2.hyd			
DME_injector_3sets.hyd			
DME_injector_inv_con.hyd			
810_Fuel Injection – FIRE Link			
SAC_CRI2_lift_ca.hyd		Hydsim file	ACCI link
SAC_CRI2_lift_dt.hyd	Primer	Hydsim file	ACCI link
SAC_CRI2_userdef_ca.hyd		Hydsim file	ACCI link
SAC_CRI2_userdef_ca.hyd		Hydsim file	ACCI link

common_rail2wv_Fire.hyd			Off line
CRIP2_injector_Fm_int.hyd			Off line
VE_pump_Fire.hyd			Off line
815_Fuel Injection – Gas			
gas_injector4.hyd			
gas_injector4_opt.hyd			
811_Fuel Injection – Gasoline			
GDI_injector1.hyd			
GDI_injector1_therm.hyd			
GDI_simple.hyd			
GDI_simple_therm.hyd			
GDI_system.hyd			
GDI_system_full.hyd			
GDI_system_simple.hyd			
HDEV5_injector.hyd			
TGDI_rail.hyd			
V8_ported_rail1.hyd			
V8_ported_rail2.hyd			
802_Fuel Injection – Heavy Oil			
fuelsystem_2inj.hyd			
Hydraulic_Cylinder_Unit_1.hyd			
injection_valve1.hyd			
injection_valves3.hyd			
Mech_pump.hyd			
820_Fuel Injection – Other			
Flow_meter_hp.hyd			
Nozzle_flow_coef.hyd			
Sac_nozzle_multicont.hyd			
Sac_nozzle_standard.hyd			
Sac_volume1.hyd			
Sac_volume1_RK5.hyd			
Sac_volume2.hyd			
Sac_volume1_RK5.hyd			
Sac_volume_gas.hyd			
Sac_volume_obsn1.hyd			
Sac_volume_obsn2.hyd			
850_General			
abs_roughness_const.hyd			
abs_roughness_var.hyd			

Bended_lines.hyd			
Contact_test.hyd			
fixed_friction_factor.hyd			
Leakage_2pistons.hyd			
Leakage_gap_comp.hyd			
Leakage_userdef_2Dtable.hyd			
Leakage_userdef_3Dtable_1.hyd			
Leakage_userdef_3Dtable_2.hyd			
local_coordinates_Cam.hyd			
local_coordinates_Rocker.hyd			
mcormack_line_1ph.hyd			
mcormack_line_1ph_fr.hyd			
mcormack_line_2ph.hyd			
mcormack_line_2ph_fr.hyd			
Orifice_std.hyd			
Orifice_std_therm.hyd			
simple_line1_ch.hyd			
simple_line1_dA.hyd	Primer		
simple_line1_gd.hyd			
simple_line1_lp.hyd			
simple_line1_mc.hyd			
simple_line2_ch.hyd			
simple_line2_dA.hyd			
simple_line2_gd.hyd			
simple_line2_lp.hyd			
simple_line2_lp_thermal.hyd			
simple_line2_mc.hyd			
simple_line_4cases.hyd			
Stop_piston_linear.hyd			
Stop_piston_nonlinear.hyd			
Stop_pseudo_linear.hyd			
Twin_piston_contact.hyd			
860_MATLAB			
common_rail2wv_dll_1.hyd	Primer		
common_rail2wv_dll_1ap.hyd			
common_rail2wv_dll_2.hyd			
common_rail2wv_dll_2ap.hyd			
comrail_4injectors_p_dll.hyd			
comrail_4injectors_pid_dll.hyd			
TGDI_rail_4inj_pid_dll.hyd			

TGDI_rail_4inj_pid_dll_2.hyd			
pressure_control_m_1.hyd	Primer		
pressure_control_m_1ap.hyd			
pressure_control_m_2.hyd			
pressure_control_m_2ap.hyd			
temperature_control_m.hyd			
Comrail2wv_abs_mdl_1.hyd	Primer		
Comrail2wv_abs_mdl_1ap.hyd			
Comrail2wv_abs_mdl_2.hyd			
Comrail2wv_abs_mdl_2ap.hyd			
Comrail2wv_abs_mdl_3case_sets.hyd			
Comrail2wv_rel_mdl_1.hyd			
Comrail2wv_rel_mdl_1ap.hyd			
Comrail2wv_rel_mdl_2.hyd			
Comrail2wv_rel_mdl_2ap.hyd			
Comrail_4injectors_mdl.hyd			
840_Mech. Drive			
camshaft_torsion.hyd			
camshaft_unit_inj4.hyd			
Elastic_shaft_1.hyd			
Elastic_shaft_2.hyd			
Lift_amplifier.hyd			
Piezo_actuator_lift.hyd			
Timing_inj_drive_4cyl.hyd			
Timing_inj_drive_4cyl_2.hyd			
830_Valve Train			
Accumulator_1.hyd			
EHVA_fingerfol_1.hyd	Primer		
EHVA_fingerfol_1el.hyd			
EHVA_fingerfol_1el_gas.hyd			
EHVA_fingerfol_2.hyd	Primer		
EHVA_fingerfol_2el.hyd			
EHVA_fingerfol_3.hyd	Primer		
EHVA_fingerfol_3el.hyd			
EHVA_fingerfol_3el_gas.hyd			
Timing_inj_drive_4cyl.hyd			
Valve_actuator.hyd			
Var_valve_train.hyd			