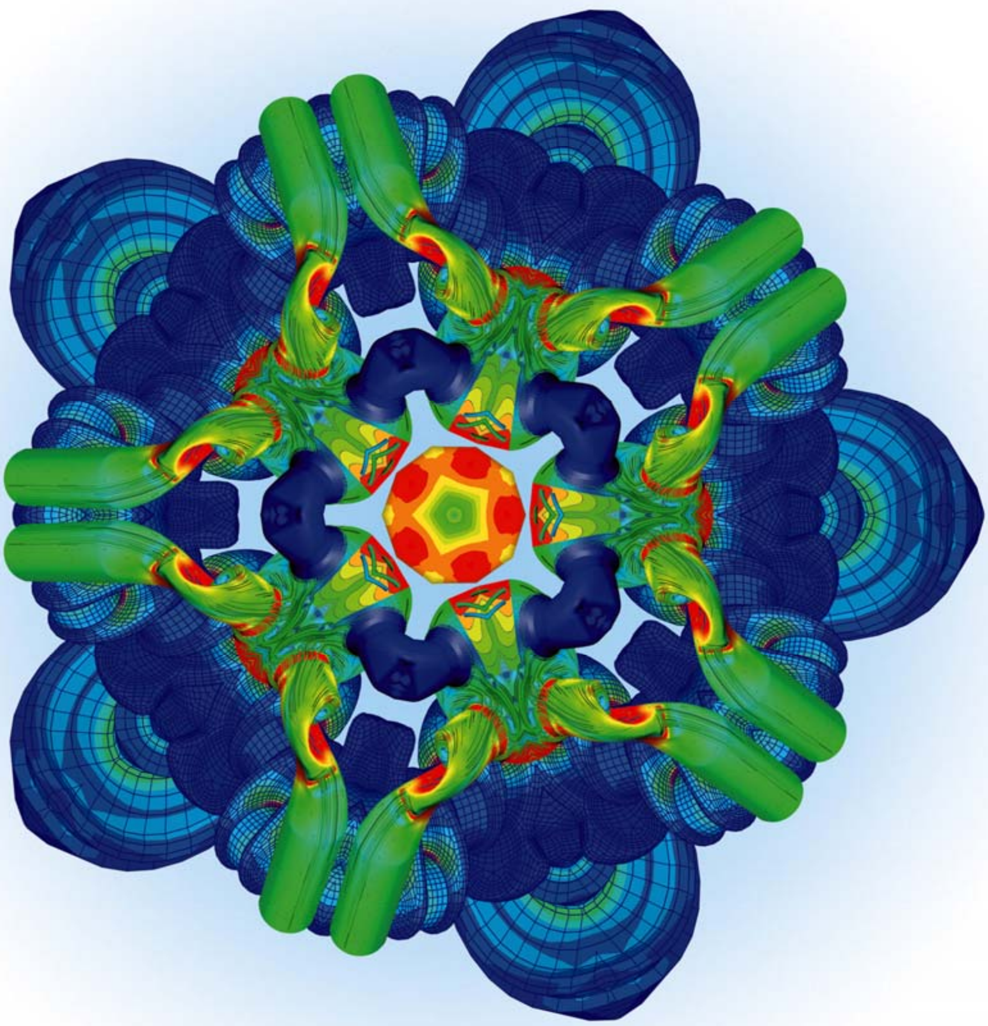
Edition 07/2013

Hydsim Users Guide

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	19-Apr-1999	HYDSIM Reference Manual V.4.1	11.0101.0530
B	25-Nov-2001	HYDSIM Reference Manual V.4.2	11.0101.0540
C	03-Mar-2003	HYDSIM Reference Manual V.4.3	11.0101.0542
D	18-Jul-2003	HYDSIM Reference Manual V.4.3.1	11.0101.0545
E	23-Jun-2004	HYDSIM Reference Manual V.4.4	11.0101.0549
F	15-Dec-2004	HYDSIM Reference Manual V.4.4.1	11.0101.0551
G	29-Jul-2005	HYDSIM Reference Manual V.4.4.2	11.0101.0554
H	09-Feb-2007	HYDSIM Users Guide v4.5	11.0101.0557
I	31-Jan-2008	HYDSIM Users Guide v4.5.4	11.0101.0560
J	27-Feb-2009	HYDSIM Users Guide v2009	11.0101.2009
K	17-Jul-2009	HYDSIM Users Guide v2009.1	11.0101.2009.1
L	02-Nov-2009	HYDSIM Users Guide v2009.2	11.0101.2009.2
M	19-Nov-2010	BOOST Hydsim Users Guide v2010	11.0101.2010
N	29-Jul-2011	BOOST Hydsim Users Guide v2011	11.0101.2011
O	02-Dec-2011	BOOST Hydsim Users Guide v2011.1	11.0101.2011.1
P	14-Jul-2013	BOOST Hydsim Users Guide v2013.1	11.0101.2013.1

This document describes how to run the BOOST Hydsim software. It does not attempt to discuss all of concepts 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 perform the dynamics analysis of hydraulic and hydromechanical 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 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. Responsibilities

This software and documents are distributed solely on an "as is" basis. The entire risk as to their quality and performance is with the user. Should either the software or this document prove defective, the user assumes the entire cost of all necessary servicing, repair, or correction. AVL and its distributors will not be liable for direct, indirect, incidental, or consequential damages resulting from any defect in the software or this document, even if they have been advised of the possibility of such damage.

1.3. User Qualifications

Users of this manual:

- Must be qualified in basic Unix, Linux or Windows systems
- Must be qualified in basic Fluid Mechanics and Dynamics
- Must be qualified in Multi-body Dynamics and Vibration

1.4. 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 to, 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 or for manual titles.
`monospace`	To indicate a command, a program or a file name, messages, input / output on a screen, file contents or object names.
SCREEN-KEYS	A **SCREEN** font is used for the names of windows and keyboard keys, e.g. to indicate that you should type a command and press the **ENTER** key.
MenuOpt	A **MenuOpt** font is used for the names of menu options, submenus and screen buttons.

1.5. Configurations

Software configurations described in this manual were in effect on the publication date of this manual. It is the user 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. Primarily, the program has been developed for the simulation of diesel injection systems. However, BOOST Hydsim is well suited for the modelling of gasoline and alternative fuel (e.g. dimethyl ether) injection systems. Furthermore, the new application such as electro-hydraulic valve trains and actuators have been successfully carried out. Moreover, the program is useful in a variety of other fields concerned with dynamic analysis of hydraulic and mechanical systems. For instance, the dynamics of hydraulic-mechanical control devices can be simulated as well as the vibration of drives. Using an interface to the MATLAB software, any MATLAB Simulink model, m-function and DLL (dynamically-linked library) can be linked to the BOOST Hydsim model.

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 figures of the physical elements) on the GUI screen. The icons can be connected by mechanical, hydraulical and/or special (logical) connections. New type of connection is the so-called wire connection which serves to link BOOST Hydsim icons to the MATLAB interface elements. GUI controls the model build-up process and does not allow incompatible connections as well as other invalid input specifications.

2.1. BOOST Hydsim Pre-Processor

The pre-processor/GUI enables the user to do the following:

- Build a 2-D representation of the BOOST Hydsim model
- Define properties and other specifications
- Generate load information
- Perform dynamic analysis
- Access IMPRESS Chart for result evaluation
- Access PP3 for animation

2.1.1. 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, each particular element of the physical system is represented by an icon (symbol) on the GUI screen. Icons contain schematic figures of the physical elements. Icons are connected by red or blue lines with arrows. Red line implies a mechanical connection (spring and/or damper) and blue line – a hydraulic connection (flow direction). Certain elements may be connected by green lines (special connections).

2.1.2. Define Properties

Once the 2D model of the system is defined, the properties of the elements and mechanical connections may be specified. For this, a selected icon has to be double-clicked with a mouse left-button. Alternatively, an input dialog box can be opened by first highlighting the icon with a mouse left-button, then opening the Pop-up Menu by clicking the right mouse button and selecting there **Properties**. In addition, by opening different dialog boxes from the Menubar, the user may specify the initial conditions, desired output parameters and define some other properties related to the element. Properties of mechanical connections (red lines) are specified in the same way. Hydraulic (blue) and special (green) connections have no user-defined properties.

2.1.3. BOOST Hydsim Execution

The BOOST Hydsim program can be run directly from the GUI by pressing one of the following buttons in the **Simulation** PullDownMenu on the Menubar. **Run** and **Run Sets** will run with 1D optimization if defined in **Search Adjust**.

- Run: Ordinary run
- Run Sets: Series calculation with Data Sets
- Restart: Restart of previously saved system

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

If no error message appears on the GUI main screen, BOOST Hydsim has started the execution of the model. The actual state of the simulation process is shown in the Simulation Status window which pops up automatically if the start procedure is successful. During numerical calculations, various information, warning, and error messages may be produced by the BOOST Hydsim kernel. These are stored on a text file simulation.out and can be viewed by pressing **View Logfile** button in the Simulation Status window or Simulation PullDownMenu. It is ultimately recommended to use this option after each start of the program (especially with a new model). Any kernel break-up caused by a fatal run-time error or data compatibility violation will be immediately reported there.

2.1.4. Post-Processor IMPRESS Chart

The IMPRESS Chart post-processor may be accessed directly from the GUI to view 2D plots of results. The IMPRESS Chart tool is used for the result evaluation of the BOOST Hydsim simulation. The results can be plotted as a function of time and (if appropriate) reference or crank angle. The desired output parameters must be selected by the user in the GUI pre-processor from a predefined list available for each element.

Each element has a predefined set of results, which (if selected by the user) are stored on a single ASCII file. By default, the data are stored on the GIDas file. The control information is stored on .ppd files. Its content is displayed in the Element Tree window integrated into the IMPRESS Chart. For optimization run, an iteration history file is also produced. Output of results is available in time domain (always), reference angle domain or crank angle domain (if relevant).

Typical simulation results for hydraulic elements are:

- Hydraulic pressure
- Temperature
- Mass or volumetric flow rate (e.g. injection rate)
- Cumulative rate (e.g. injection quantity)
- Hydraulic force
- Cross-sectional flow area
- Flow resistance /flow discharge coefficients
- Cavitation factors

For mechanical elements, typical simulation results would be:

- Coordinate, velocity and/or acceleration
- Mechanical forces and torques
- Kinematic parameters

The post-processing of the data (plots) is carried out by IMPRESS Chart. It allows a flexible automated generation of plots by use of predefined templates (delivered with BOOST Hydsim or designed by the user) as well as the interactive creation of graphs, diagrams, etc.

2.1.5. Post-Processor PP3

The PP3 post-processor is a 3-D animation tool. It may also be accessed from the GUI or directly from BOOST Hydsim **Simulation** PullDownMenu, by opening Nozzle Flow window with **Animation | Nozzle Flow** command and clicking respective **Show** button there. The PP3 tool is used for the nozzle flow and spray animation. To run it successfully, BOOST Hydsim simulation must be performed beforehand. Basic and Extended SAC and VCO Nozzle elements can be animated. For conventional injection systems, animation displays needle motion, pressure variation in nozzle bore, nozzle chamber, sac volume (if appropriate), needle back pressure, leakage through needle guide, spray angle, penetration and spray intensity (if calculated). For common rail systems, control piston motion and control volume pressures are included into animation.

2.2. Model Structure

The system model for BOOST Hydsim may contain various hydraulic, mechanical and general-purpose elements. Elements are joined into groups according to their type and functionality. In this way, BOOST Hydsim has sixteen element groups which names are listed in the Element Menu of the Workspace window. These are as follows:

Boundary group

- Pressure/Temperature
- Flow Rate
- Mechanical
- Hydromechanical

Cam group

- Cam Profile (lift or acceleration data)
- Cam Plate (lift or acceleration data) *

Lever group

- Rocker Arm (Pushrod-actuated) *
- Rocker Arm (Cam-actuated) *
- Finger Follower *

Solid group

- Lumped Mass
- Rigid Shaft

Piston group

- Standard
- Split-Injection Device (SID) *

Volume group

- Standard
- Compliant
- Two-Phase *

Line group

- d'Alembert Model
- Laplace Transform
- Characteristics Method *
- Godunov Method *
- MacCormack/Two-phase

Bend group

- Round/Circular *
- Mitre/Circular *

Junction group

- Tee (90 deg)
- Tee (angle)

Pump group

- Radial Piston Distributor Pump (RPD) *
- Plunger

* Currently the Variable-step solver does not support these elements.

- Ejector (Hydraulic)

Leakage group

- Annular Gap

Port group

- In-line Fill/Spill
- Distributor Fill/Spill

Valve group

- Delivery
- Constant Volume
- Check (Ball)
- Check (Poppet)

Throttle group

- Time-controlled (Switch valve)
- Lift-controlled (Slide valve)
- Flow Area vs. Time/CA Function
- Pressure vs. Flow Rate Function

Solenoid group

- Armature (Basic model)
- Armature (Extended model)

Piezo group

- Lift Function
- Lift Amplifier *
- Stack Actuator *

Orifice group

- General
- Cavitating
- Sharp-edged
- Round-edged
- Sharp-edged Long

Nozzle group

- Sac Nozzle Orifice (Basic model)
- Valve-covered Nozzle Orifice (Basic model)
- Sac Nozzle Orifice (Extended model)
- Valve-covered Nozzle Orifice (Extended model)
- RSN-Collar Throttle (Rate-shaping Nozzle) *
- CRI3-Guide Flow *

Needle group

- Standard

- 2-spring

Gas group

- Pressure/Temperature
- Volume
- Lift -controlled Throttle
- Flow Area vs. Time/CA

Control group

- API: Simulink *
- API: m-function *
- DLL (dynamically-linked library) *
- PID Controller

External group

- FIRE Link *
- BOOST Link - Valve *
- BOOST Link - Plenum *

By double clicking on the group with the left mouse button, the entire list of elements belonging to that group appears. By double clicking on the selected element the corresponding icon is placed into the model window. The complete group and their element list are given in Chapter 24.

2.3. General Input Data Descriptions

The Input Data Dialog Boxes can be accessed in three ways:

- Double click on the element
- Highlight the element. From the PullDownMenu, **Element | Properties**
- Click on the element with right mouse button and choose **Properties** from the PullDownMenu.

Element name	Arbitrary text is specified to enable easier identification of the element. It is stored as a headline (up to the first 12 characters) in the corresponding output files. With `.ppd` file (Post-processor **IMPRESS Chart Results | Model Tree**), the full name is used. With GIDas file (**IMPRESS Chart Results | Load GIDas File**), up to 12 first characters are used (including spaces). With DATa file, up to first 8 characters will be used.
Data File	Specify data file from which the data will be loaded. This box becomes active when the Automatic Reload button is active.
Automatic Reload	If the Automatic Reload button is active, the file is automatically reloaded when the calculation is started. Automatic Reload button will not be faded out after table is loaded from file.
Use Default Data File Path	If the button Use Default Data File Path is active, the path name always starts in the directory where the AWS was started (all directory information is deleted from the Data File box, only the file name remains).

2.3.1. TABLES

The following are general rules for the specification of table data:

1. The intermediate values are bridged by linear interpolation.
2. If the actual angle exceeds the maximum value in the table, the last specified value of polar coordinate is used for the further calculation.
3. If the actual angle is less than the angle specified in the first table row, the first specified value of polar coordinate will be used.
4. The data can be entered:
   - Directly by typing them into the corresponding columns. Highlight the input field by clicking it twice, type the desired value and press return. New row can be introduced by pressing the **Insert row** bar.
   - From a file by pressing the **Load** bar. After this a file selection window is shown, where the file name is specified.
5. Rows can be removed by highlighting one value and pressing the **Remove row** bar.
6. A table can be cleared by pressing the **Clear All row** bar.
7. Dependence of effective cross-sectional flow area or volumetric flow rate on needle lift can be viewed in IMPRESS Chart window by pressing the **View** bar.
8. The specified table can be stored on separate file by pressing the **Store bar.** Then a file selection window is opened where the output file name is to be given.

3. BOUNDARY

Boundary elements in BOOST Hydsim are used for defining the boundary conditions of the system. Boundary group contains the following elements:

- **Pressure/Temperature Boundary (liquid)**
- **Flow Rate Boundary**
- **Mechanical Boundary**
- **Hydro-mechanical Boundary**

☞ **Note:** **Gas Pressure/Temperature Boundary** is located in **Gas Boundary** group.

General rules for the specification of boundary conditions (**Input Dialog** windows):

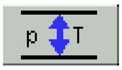
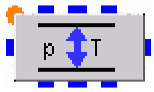
- Intermediate values are bridged by linear interpolation.
- If the defined region (Time, Reference angle) for the boundary condition starts later than the calculation, the boundary values specified in the 1st row are used.
- If a boundary condition is kept constant throughout the calculation, it has to be specified only once (for the entire calculation domain).
- If the calculation exceeds the defined region the following two cases occur:

  If the Copy data button is not active for each cycle, the last specified value is kept constant for further calculations. This will be the case if in the **Calculation Control** dialog box, a longer Time (Reference angle) interval is specified than for the boundary condition.

  If the Copy data button is active for each cycle and if more than one cycle is considered in the calculation, it is sufficient to define the values for one cycle. The program will copy data from the table for each new cycle.

☞ **Note:** If the Copy data button is **active** for each cycle in global dialog box Simulation/Boundary Data then it will have dominant status and it will be active at Simulation/Run, Run Sets or Restart.

3.1. Pressure/Temperature Boundary

Element Name:	**Pressure/Temperature Boundary**	
Element Icon:		
Definition:	This element serves to define the pressure and temperature on the outer connections (boundaries) of the system as functions of time or reference angle.	
Connecting pins:	standard pins: 8 (all hydraulic) special pins: 0 wire pins: 1 Pressure/temperature is the same at all pins.	

Note: *x-direction* is the only hydraulic direction.

3.1.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Shift Time/Angle relative to calculation start	Specify the Shift time/angle for the first table column. It will be added to the time/angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Copy data for each cycle (360 deg Ref. angle):	Activation enables specification of the values in the table as a function of time (or reference angle) only for one cycle (360 degrees of Reference Angle). If more than one revolution of the cam is considered in the calculation, the program will copy data from the table for each new cycle.
Pressure and Temperature vs. Time/Reference Angle:	Specify the Pressure and/or Temperature as a function of time (or reference angle) in form of table.

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **8.1.8**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)

3.1.2. Initial Conditions

No initial conditions can be specified for **Pressure/Temperature Boundary** element.

Pressure and temperature values for time/angle at calculation start (refer to **Calculation Control** dialog in section **26.2.6.3**) are taken as initial conditions for the connected hydraulic elements (**Piston type, Orifice type, Leakage and Lines**).

3.1.3. Modify Parameter

Pressure/Temperature Boundary has no modifiable parameters.

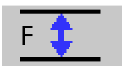
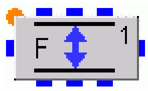
3.1.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box on the left of the parameter name has to be checked.

Pressure	Actual pressure in **Pressure/Temperature Boundary**.
Temperature	Actual temperature in **Pressure/Temperature Boundary**.
For **Pressure/Temperature Boundary** element, the input and output data is the same.	

3.2. Flow Rate Boundary

Element Name:	**Flow Rate Boundary**	
Element Icon:		
Definition:	This element serves to define the flow rate on outside connections (boundaries) of the system as functions of time or reference angle.	
Connecting pins:	standard pins: 8 (all hydraulic) special pins: 0 wire pins: 1 Flow Rate is same at all connecting pins.	

Note: *x-direction* is the only hydraulic direction.

3.2.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Shift Time/Angle relative to calculation start	Specify the Shift time/angle for the first table column. It will be added to the time/angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Copy data for each cycle (360 deg Ref. angle):	Activation enables specification of the values in the table as a function of time (or reference angle) only for one cycle (360 degrees of Reference Angle). If more than one revolution of the cam is considered in the calculation, the program will copy data from the table for each new cycle.
Flow Rate vs. Time/Reference Angle:	Specify the Flow Rate as a function of time (or Reference angle) in form of table.

3.2.2. Initial Conditions

No initial conditions can be specified for **Flow Rate Boundary** element.

Flow rate values for time/angle at calculation start (refer to **Calculation Control** in section 26.2.6.3) are taken as initial conditions for the connected **Volume** elements.

3.2.3. Modify Parameter

Flow Rate Boundary has no modifiable parameters.

3.2.4. Output Parameters

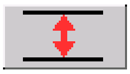
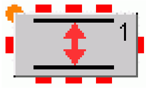
Path: **Element | Store Results**

To activate output parameter, the check box on the left of the parameter name has to be checked.

Description of output parameters:

Volumetric Flow Rate	Volumetric Flow Rate from **Flow Rate Boundary** in **Time** domain.
Volumetric Flow Rate	Volumetric Flow Rate from **Flow Rate Boundary** in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter **26.2.6.3**). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
For **Flow Rate Boundary** element, the input and output data are the same.	

3.3. Mechanical Boundary

Element Name:	**Mechanical Boundary**	
Element Icon:		
Definition:	This element serves to define the coordinate or velocity on outside connections (boundaries) of the system as functions of time or reference angle.	
Connecting pins:	standard pins: 8 (all mechanical) special pins: 0 wire pins: 1	
	Coordinate and velocity are same at all connecting pins.	

Note: One **Mechanical Boundary** element defines coordinate or velocity in only one direction. If it is necessary to specify mechanical boundary conditions for more than one direction, a separate **Mechanical Boundary** has to be defined for each direction and connected via mechanical connections to desired element (*x*, *y*, and *w* direction are possible).
It is not possible to specify both coordinate and velocity within same **Mechanical Boundary** element. If coordinate is defined in input table, velocity in same direction will be obtained by time differentiation. If velocity is defined in input table, coordinate in same direction will be obtained by time integration. Coordinate/Velocity pairs will be calculated for specified direction in time domain.

3.3.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Variable:	This button selects coordinate or velocity in the required direction. The active variable button will also be the 2nd column title in the **Coordinate/Velocity** table.
Shift Time/Angle relative to calculation start	Specify the Shift time/angle for the first table column. It will be added to the time/angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started
Copy data for each cycle (360 deg Ref. angle):	Activation enables specification of the values in the table as a function of time (or reference angle) only for one cycle (360 degrees of Reference Angle). If more than one revolution of the cam is considered in the calculation, the program will copy data from the table for each new cycle.
Coordinate/ Velocity:	

Units for different variables:

X-coordinate	Unit:	Simulation Period Units / Length Units
Y-coordinate	Unit:	Simulation Period Units / Length Units
W-coordinate (angle)	Unit:	Simulation Period Units / Angle Units
X-velocity	Unit:	Simulation Period Units / Velocity Units
Y-velocity	Unit:	Simulation Period Units / Velocity Units
W-angular velocity	Unit:	Simulation Period Units / Angular Velocity Units
Direction of Element motion		
x(e1) **y(e2)** **z(e3)**	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).	
LINK TO EXCITE POWER UNIT		
Element reaction force on housing is calculated (refer to section **0**).		
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)	
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)	
Mechanical boundary elements are always connected with other mechanical elements via the mechanical connection. The mechanical connections represent springs and dampers between elemenets. Any deformation of spring or velocity difference in thedamper will cause forces on the boundary and they will be calculated.		

3.3.2. Initial Conditions

No initial conditions can be specified for **Mechanical Boundary** element.

Coordinates or velocities values for time/angle at calculation start (refer to **Calculation Control** in section 26.2.6.3) are taken as initial conditions.

3.3.3. Modify Parameter

Mechanical Boundary has no modifiable parameters.

3.3.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box on the left of parameter name has to be checked.

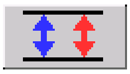
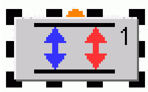
Description of output parameters:

x-coordinate	x-coordinate of **Mechanical Boundary**.
Velocity in x-direction	Velocity in *x*-direction of **Mechanical Boundary**.
y-coordinate	*y*-coordinate of **Mechanical Boundary**.
Velocity in y-direction	Velocity in *y*-direction of **Mechanical Boundary**.
w-coordinate (rotation)	*w*-coordinate (rotation angle) of **Mechanical Boundary**.
velocity in w-direction	Angular velocity of **Mechanical Boundary** around rotation axis *w*.

For **Mechanical Boundary** element, the input and output parameters are the same.

Note: Do not activate output parameters in directions which are not specified in **Input Data** dialog box.

3.4. Hydromechanical Boundary

Element Name:	**Hydromechanical Boundary**	
Element Icon:		
Definition:	This element serves to define 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.	
Connecting pins:	standard pins: 10 (all general-purpose) special pins: 0 wire pins: 1 Coordinate, velocity (pressure and flow rate) are same at all connecting pins.	

Note: Once activated, a general-purpose pin acquires a specific connection type (mechanical or hydraulic) which cannot be changed afterwards.

One **Hydromechanical Boundary** element defines coordinate or velocity in only one direction. If it is necessary to specify mechanical boundary conditions for more than one direction, a separate **Mechanical Boundary** has to be defined for each direction and connected via mechanical connections to the desired element (*x*, *y* ,and *w* directions are possible).

It is not possible to specify both coordinate and velocity within same **Hydromechanical Boundary** element. If coordinate is defined in input table, velocity in same direction will be obtained by time differentiation. If velocity is defined in input table, coordinate in same direction will be obtained by time integration.

Coordinate/Velocity pairs will be calculated for specified direction in time domain.

☞ **Note:** *X-direction* is the only hydraulic direction. If coordinate or velocity in *x-direction* is specified, additional values for Pressure and Flow can be given as these are independent variables. If coordinates or velocities in *y* or w directions are specified, no Pressure or Flow can be defined (respective columns are greyed out).

If **Hydromechanical Boundary** has a hydraulic connection to the neighbouring element, only pressures and/or flow will be transferred to it. However, in case of mechanical connection, all defined variables (coordinate/velocity, pressure and/or flow) will be transmitted to the connected element (**exception to general rule**).

In other words, if **Hydromechanical Boundary** element is attached via mechanical connection to the element which allows both types of connections (mechanical and hydraulic, as **Piston** element), pressure defined in **Boundary** table will act on piston connection-area and produce hydraulic connection force, even if there is no hydraulic connection.

3.4.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Variable:	This button selects coordinate or velocity in the required direction. The active variable button sets the column head in the **Table of Boundary Conditions** table.
Shift Time/Angle relative to calculation start	Specify the Shift time/angle for the first table column. It will be added to the time/angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for the first column of the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Copy data for each cycle (360 deg Ref. angle):	Activation enables specification of the values in the table as a function of time (or reference angle) only for one cycle (360 degrees of Reference Angle). If more than one revolution of the cam is considered in the calculation, the program will copy data from the table for each new cycle.
Table of Boundary Conditions:	

Units for different variables:

X-coordinate / Pressure / Flow Rate	Unit:	Simulation Period Units / Length Units / Pressure Units / Volume Flow Units
Y-coordinate	Unit:	Simulation Period Units / Length Units

W-coordinate (angle)	Unit:	Simulation Period Units / Angle Units
X-velocity / Pressure / Flow Rate	Unit:	Simulation Period Units / Velocity Units / Pressure Units / Volume Flow Units
Y-velocity	Unit:	Simulation Period Units / Velocity Units
W-angular velocity	Unit:	Simulation Period Units / Angular Velocity Units

Specify the Coordinate, Velocity, Pressure, and Flow Rate as a function of time or Reference angle in form of table.

Direction of Element motion	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated.	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Hydromehcanical elements are connected with other mechanical elements via the mechanical connection. The mechanical connections represent springs and dampers between elemenets. Any deformation of spring or velocity difference in the damper will cause forces on the boundary and they will be calculated. Hydromehcanical elements are pressurized with boundary pressure and hydraulic force acts on them (refer to section **8.1.8**).	

3.4.2. Initial Conditions

No initial conditions can be specified for **Hydromechanical Boundary** element.

Values for time/angle at calculation start (refer to **Calculation Control** in section 26.2.6.3) are taken as initial conditions for the connected elements according to the following rule:

- Flow rate: for **Volume** elements only
- Pressure: for other hydraulic elements (**Piston type, Orifice type, Leakage and Lines**).

3.4.3. Modify Parameter

Hydromechanical Boundary has no modifiable parameters.

3.4.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box on the left of parameter name has to be checked.

Description of output parameters:

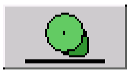
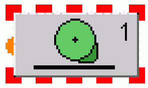
x-coordinate	*x*-coordinate of **Mechanical Boundary**.
velocity in x-direction	Velocity in *x-direction* of **Mechanical Boundary**.
Pressure	Actual pressure in **Hydraulic Boundary**.
Volumetric Flow Rate /time	Volumetric Flow Rate from **Hydraulic Boundary** in Time domain.
Volumetric Flow Rate /angle	Volumetric Flow Rate from **Hydraulic Boundary** in Reference angle domain or Crank angle domain. **Note**: Output angle domain is defined in Calculation Control input dialog box (refer to 26.2.6.3). **Note**: Output values will be 0, if the **Time** domain button is active in Calculation Control dialog box.
y-coordinate	*y*-coordinate of **Mechanical Boundary**.
velocity in y-direction	Velocity in *y-direction* of **Mechanical Boundary**.
w coordinate (rotation)	*w*-coordinate (rotation angle) of **Mechanical Boundary**.
velocity in w-direction	Angular velocity of **Mechanical Boundary** around rotation axis *w*.

For **Hydromechanical Boundary** element, the input and output parameters are the same.

☞ **Note:** Do not activate output parameters in directions which are not specified in **Input Data** dialog box.

4. CAM

4.1. Cam Profile

Element Name:	**Cam Profile**
Element Icon:	
Definition:	This element serves to define the Cam Profile by follower acceleration or lift data (grinding coordinates for reciprocating follower).
Connecting pins:	standard pins: 10 (all mechanical) special pins: 0 wire pins: 1 

Note: If there are mechanical connections on input side (cam center), it must be specified in all three directions.
x and *y-directions* always denote translational motion (e.g. camshaft bending), therefore only translational springs and/or dampers can be attached there.
Direction *w* is associated with rotational motion and requires torsional spring and/or damper to be connected.
On output side (cam profile), mechanical connection can be specified only in *x-direction* (refer Figure 1).

Note: For **Cam Profile** element, input parameters and initial conditions have to be defined in local x-y coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: **Cam Profile** element local x-y coordinate system may be **translated** and **rotated** global x-y coordinate system. Rotation is defined through parameter: Direction of Follower motion.

Note: **Cam Profile** acceleration/lift data implies the acceleration/lift of the roller center of the reciprocating **Cam Profile** follower, provided the connection between the cam profile and the roller is rigid. If there is e.g. a rocker arm in between, the **Cam Profile** acceleration/lift has to be recalculated according to the rocker arm geometry (or better to use another element **Rocker Arm** (**Cam**)).

Mechanical model of **Cam Profile** with possible connections is Figure 1.

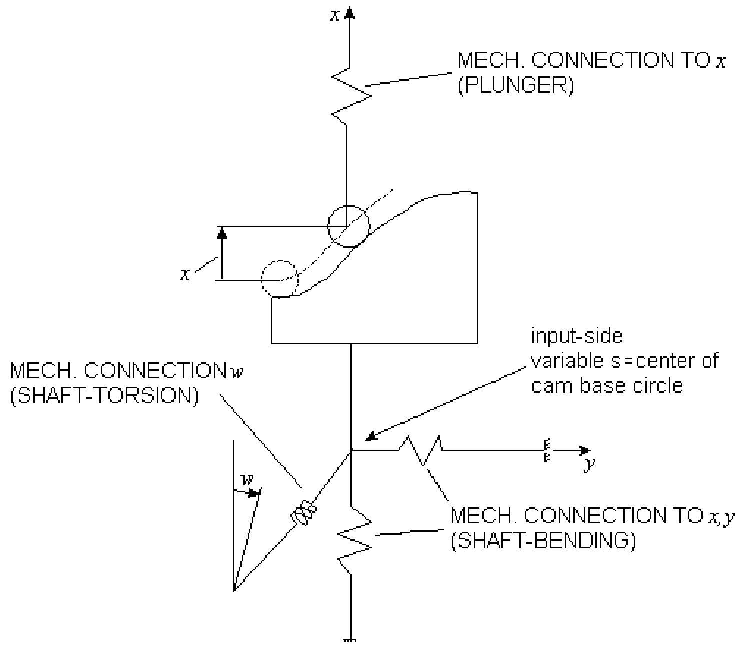


Figure 1: Mechanical Model of Cam Profile

4.1.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Transmission ratio vs. Reference speed	Specify transmission ratio between cam rotation speed and Reference speed in **Calculation Control** dialog. Cam speed is calculated as Reference speed multiplied with transmission ratio. **Note:** If an input connection in *w*-direction exists (from e.g. **Shaft**), then Transmission ratio input is disabled. In this case speed of connected element (**Shaft**) is used as **Cam** speed.
Radius of cam base circle	Specify the radius of cam base circle of **Cam Profile**. The radius of cam base circle corresponds to R_{base} in Figure 2.
Radius of roller	Specify the radius of roller. The roller radius corresponds to r_{roll} in Figure 2.
Follower eccentricity (offset)	Specify the eccentricity of the follower axis with respect to the cam center.
The radii of cam base circle and roller are required for the calculation of torsional shaft distorsion, pressure angle, reaction force on the cam profile, shaft torque, etc. For dynamic calculation alone with **Cam Profile** being a boundary element, they are not necessary. However, they are absolutely necessary if a **Shaft** or other element is connected to the **Cam Profile** input end.	
Follower lift at calculation start	Active only for **Cam Profile** defined in terms of the follower acceleration data. Specify the follower lift at calculation start. This will be used internally as constant of integration for calculation of initial follower velocity.

Initial follower velocity at 1000 rpm	Specify the follower velocity at calculation start with cam speed 1000 rpm, which is used for internal calculation of initial follower lift or acceleration. For cam speed other than 1000 rpm, initial follower velocity has to be recalculated by linear proportion.
CAM PROFILE	
Shift angle relative to calculation start	Specify the shift angle of cam profile relative to the start position. Positive angle 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, the initial conditions of the follower must be equal to zero. Negative shift angle implies the calculation start from the cam profile value corresponding to the shift angle value in the cam profile table. In this case, the appropriate number of rows in the table will be skipped until the row with the cam angle equal to shift angle is found. Clearly, initial conditions of the follower motion for the negative shift angle must be nonzero. Shift angle must be defined in the range 0…360 degrees.

☞ **Note:** If the sum of the cam profile angle interval with shift angle exceeds 360 degrees, initial conditions of the follower motion have to be non-zero.

Note: Negative shift angle is not implemented in BOOST Hydsim.

Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in the 1st column will be multiplied with the scale factor as soon as the calculation is started.
Lift of reciprocating cam follower	Click to activate lift of reciprocating cam follower in 2nd column.
Follower acceleration at 1000 rpm	Click to activate follower acceleration at 1000 rpm in 2nd column.
Lift, velocity and acceleration at 1000 rpm	Click to activate follower lift, velocity and acceleration at 1000 rpm in 2nd, 3rd and 4th columns, respectively.

☞ **Note:** Follower lift as input data standalone must be used with extreme caution. If the lift data is not precise enough, the numerical derivatives (velocity and acceleration) may contain coarse errors, and calculation results will be incorrect. The follower acceleration curve must be checked carefully within each calculation.

Cam profile table	Cam profile has to be specified for the entire calculation interval (End Reference angle) given in the **Calculation Control** dialog. However, if more than one revolution of the cam is considered in the calculation, it is sufficient to define the cam profile for one revolution of the camshaft (360 degrees). Intermediate points of the cam profile are calculated by linear interpolation.

Cam profile can be viewed in Cartesian coordinates by pressing the **View** bar. If pressed, a reduced Impress Chart window is opened where the cam follower acceleration/lift curve is displayed.

HERTZ STRESS CALCULATION	
Young's modulus of cam material	Specify the Young's modulus of cam material. It corresponds to E_{cam} in formula for Hertz stress calculation (refer to Equation 5).
Effective roller (contact) width	Specify the effective width of roller. It corresponds to b_{roll} in formula for Hertz stress calculation (refer to Equation 5).
DIRECTION OF Follower MOTION	
x (e1)	Specify the unit vector of motion for the follower element in the shaft (global) coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the shaft (global) coordinate system, e_1, e_2, e_3 are components of the follower unit vector in the global *x*, *y* and *z* direction (see Figure 4). The direction of the unit vector of the follower motion corresponds to the direction of cam *x*-axis.
y (e2)	
z (e3)	
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated.	
Element (separate force)	The calculated fsorce is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force in x and y directions ($F_x = F_{norm}\cos(\beta + \Delta\beta), F_y = F_{norm}\sin(\beta + \Delta\beta)$) only if there is no input mechanical connections.	

Note: Direction of follower motion in **Cam Profile** element must correspond to direction of motion of element (**Plunger**) connected to output end of **Cam Profile**.

Note: BOOST Hydsim mechanical part is based on theory of vibration of 2D multi-body systems (translation in *x* and *y* and rotation around *w*). Therefore 3rd translational coordinate *z* is inactive (grayed out) in the input dialog.

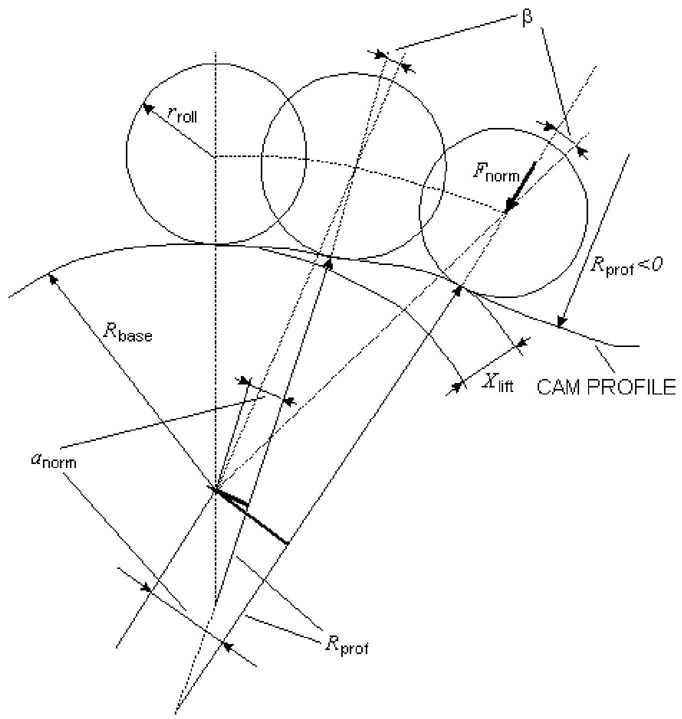


Figure 2: Geometry of Cam Profile

4.1.2. Initial Conditions

Path: **Element | Initial Conditions**

All initial values that are not specified in this dialog are set to 0.

Table 1: Initial Conditions of Cam Profile

INITIAL TRANSLATION	
x-coordinate	Specify the initial position of cam center in local *x*-direction.
Velocity in x-direction	Specify the initial velocity of cam center in local *x*-direction.
y-coordinate	Specify the initial position of cam center in local *y*-direction.
Velocity in y-direction	Specify the initial velocity of cam center in local *y*-direction.
INITIAL ROTATION	
w-angle	Specify the initial angle of cam base body in *w*-direction.
Angular velocity in w-direction	Specify the initial angular velocity of cam base body rotation in *w*-direction.

☞ **Note:** Initial angular velocity of **Cam Profile** has to be specified as follows: if **Cam Profile** has no input connection in w-direction, then set initial velocity to Reference speed multiplied by transmission ratio. If **Cam Profile** has an input mechanical connection in w-direction (e.g. from **Shaft** element), then set initial velocity to Reference speed (defined in **Calculation Control**).

☞ **Note:** BOOST Hydsim may encounter convergence problems during the calculation of torsional distortion if e.g. a **Shaft** element is connected on input end.

4.1.3. Modify Parameter

Cam Profile element has no modifiable parameters.

4.1.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box on the left of parameter name has to be checked. Description of output parameters is given in Table 2.

Table 2: Output Parameters of Cam Profile

x-coordinate of cam center	*x*-coordinate of cam center (refer to Figure 4). Nonzero only if **Cam Profile** is attached via input mechanical connections to e.g. **Mass** or **Shaft** elements (i.e. **Cam Profile** is not a boundary element).
y-coordinate of cam center	*y*-coordinate of cam center (refer to Figure 4). Nonzero only if **Cam Profile** is attached via input mechanical connections to e.g. **Mass** or **Shaft** elements (i.e. **Cam Profile** is not a boundary element).
w-coordinate of cam body (rotation angle)	*w*-coordinate (rotation angle) of cam base body. If **Cam Profile** is a boundary element (i.e. an input mechanical connection in *w*-direction does not exist), rotation angle of cam profile is the product of Reference speed $\dot{\varphi}_{ref}$, transmission ratio i_{tr} and time interval from the beginning of calculation t: $$w = \dot{\varphi}_{cam}\, t = i_{tr}\dot{\varphi}_{ref}\, t\ , \quad (\ )$$ If an input mechanical connection in *w*-direction exists, then rotation angle is calculated from the dynamics of the system.
angular velocity of cam body rotation	Angular velocity in *w*-direction of cam base body rotation. If **Cam Profile** is a boundary element (i.e. it does not have any input mechanical connection in *w*-direction), its angular velocity is equal to Reference speed multiplied by transmission ratio. Otherwise angular velocity of cam profile body is calculated at each step from the equations of motion of the system.
x-coordinate of cam profile	Local x-coordinate of **cam profile** at its contact point with the follower.
velocity of cam profile in x-direction	Velocity in local *x*-direction of **cam profile** at its contact point with the follower.

☞ **Note:** x-coordinate of cam profile and velocity of cam profile in x-direction, *x*-coordinate and velocity of output (Cam Profile) is not identical to the x-coordinate and velocity of cam follower (e.g. Plunger) because there is a mechanical connection in between. If it is very stiff (almost rigid), both coordinates and velocities will be practically equal.

velocity of cam profile in x-direction at 1000 rpm	Velocity of **cam profile** contact point with the follower in local x-direction at 1000 rpm. It is obtained by numerical differentiation of lift or integration of acceleration data defined in **Input Data** dialog box.
cam profile acceleration in x-direction at 1000 rpm	Acceleration of **cam profile** contact point with the follower in local x-direction at 1000 rpm. It is obtained by numerical differentiation of velocity or from acceleration data defined in **Input Data** dialog box.
pressure angle of cam profile	Pressure angle of **Cam Profile**. It is an angle between the normal force on **cam profile** and straight line connecting cam center to roller center.
amplitude of cam center	Amplitude of cam center is calculated from co-ordinates of cam center in x and y-direction $A_{cc} = \sqrt{x_{cc}^2 + y_{cc}^2}$. (1) It is nonzero only if mechanical connections between **Cam Profile** and e.g. **Shaft** exists in x and/or y-direction as shown in Figure 3.
torsional distortion angle	Torsional distortion angle corresponds to $\Delta\beta$ in Figure 3. It is used to correct the cam angle in order to maintain permanent contact between cam contour and follower (roller). $\Delta\beta = \alpha_{tabl} - w,$ (2) where α_{tabl} is the angle in 1st column of **Cam Profile** table and *w* is rotation angle of the cam body (camshaft). $\Delta\beta$ is equal to 0, if input mechanical connection does not exist.
reaction force normal to cam profile	Resistance force normal to cam profile F_{norm} (refer Figure 2). It is calculated from the reaction force of the cam follower (e.g. plunger) in x-direction F_x according to the formula: $F_{norm} = \dfrac{F_x}{cos(\beta + \Delta\beta)}$ (3)
torque of normal force about rotation axis *w*	Cam body torque caused by force F_{norm} which acts at an angle β to *x*-direction (refer to Figure 2). For a cam without offset: $T_{cp} = F_{norm} \sin\beta \cdot (R_{base} + x_{lift} + r_{roll}),$ (4) where x_{lift} is cam profile lift in *x*-direction.
torsional vibration velocity	Angular velocity of cam body relative to stationary rotation. Torsional vibration velocity is the difference between actual angular velocity in *w*-direction and reference speed, which is defined in **Calculation Control** dialog box. It is nonzero only if an input mechanical connection in w-direction exists.
radius of curvate of cam profile	Radius of curvature of cam profile corresponds to radius R_{prof} (refer to Figure 2).

Hertz stress at cam-roller contact	Hertz stress at cam-roller contact is calculated from the formula $$p_{Hertz} = 0.418\sqrt{\frac{E_{cam}F_{norm}}{b_{roll}}\left(\frac{1}{R_{prof}}+\frac{1}{r_{roll}}\right)}. \quad (5)$$
maximum Hertz stress till actual point	Maximum Hertz stress value from calculation start till actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).

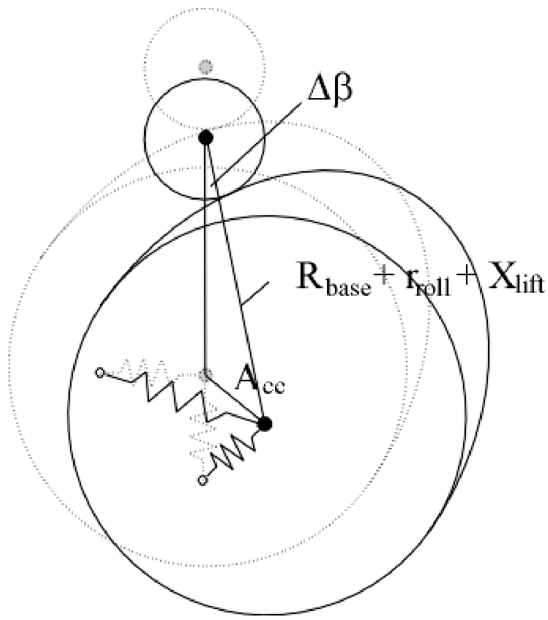


Figure 3: Amplitude of Cam Center (A_{cc}) and Torsional Cam Distortion ($\Delta\beta$)

4.1.5. Additional Information

To connect **Cam Profile** to a **Shaft** element three mechanical connections (in x, y, and w directions, respectively) are necessary as shown in Figure 4.

- To denote a rigid (or almost rigid) connection between the cam and shaft, the respective connections must possess very high stiffness (e.g. comparable to the shaft bearing stiffness).
- **Cam Profile** has three degree of freedom (refer to Figure 4). Equations governing the cam body motion are discussed in Section 6.
- If **Cam Profile** does not have input mechanical connections, co-ordinates and velocities of cam center in x and y-directions will be 0.
- **Cam Profile** has no mass and moment of inertia. Its mass and moment of inertia can be added to the mass and moment of inertia to the connected **Shaft** element.
- Preload, stiffness and damping of input mechanical connections from **Shaft** element in x and y-directions have to be defined in global coordinate system. This is important at the nonzero angle between local (**Cam Profile**) and global (**Shaft**) coordinate systems as shown in Figure 4.

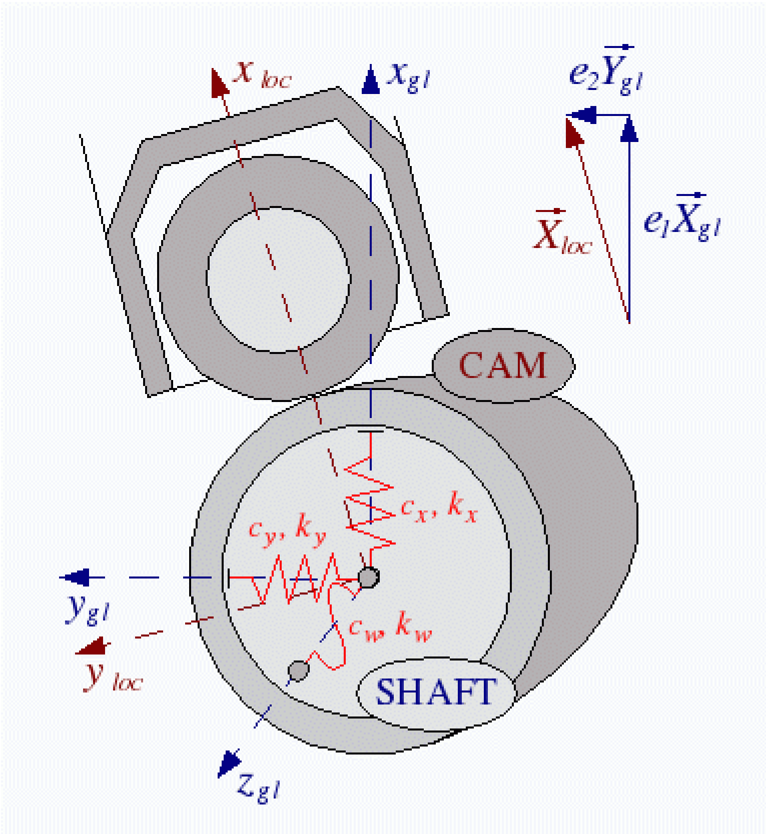


Figure 4: Cam Profile with connections to Shaft

4.1.6. Equations of Motion

Local coordinate system of **Cam Profile** and connected elements (with their connections) can have only translational motion with respect to each other and the global coordinate system. The motion of **Cam Profile** is governed by the following equation:

$$\sum_{i=1}^{n_x} c_{xi}(\dot{x}-\dot{x}_i)-\sum_{j=1}^{l_x} c_{xj}(\dot{x}-\dot{x}_j)+\sum_{i=1}^{n_x} k_{xi}(x-x_i)-\sum_{j=1}^{l_x} k_{xj}(x-x_j)=\sum_{i=1}^{n_x} F_{0xi}-\sum_{j=1}^{l_x} F_{0xj},$$

$$\sum_{i=1}^{n_y} c_{yi}(\dot{y}-\dot{y}_i)+\sum_{i=1}^{n_y} k_{yi}(y-y_i)=\sum_{i=1}^{n_y} F_{0yi}+F_y, \qquad (6)$$

$$\sum_{i=1}^{n_w} c_{wi}(\dot{w}-\dot{w}_i)+\sum_{i=1}^{n_w} k_{wi}(w-w_i)=\sum_{i=1}^{n_w} T_{0i}+T_{Fx},$$

where

x and y	coordinates of the center of cam base circle
w	rotation angle of cam body
x_i, y_i and w_i	coordinates and rotation angle of *i-th* connected elements on input end
x_j	coordinate of the *j-th* connected element on output end

c_{xi} and c_{xj}	damping coefficients (functions) of the *i-th* input and *j-th* output mechanical connections in *x-direction*
k_{xi} and k_{xj}	stiffness coefficients (functions) of the *i-th* input and *j-th* output mechanical connections in *x-direction*
c_{yi} and k_{yi}	damping and stiffness coefficients (functions) of the *i-th* input mechanical connection in *y*-direction
c_{wi} and k_{wi}	torsional damping and stiffness coefficients (functions) of the *i-th* input mechanical connection in *w*-direction
F_{0xi} and F_{0xj}	preload forces of the *i-th* input and *j-th* output mechanical connections in *x*-direction
F_{0yi}	preload force of the *i-th* input mechanical connection in *y*-direction
F_y	force in *y*-direction caused by the reaction force normal to cam profile
T_{0i}	preload torque of the *i-th* input mechanical connection in *w* direction
T_{Fx}	resistance torque of the reaction force normal to cam profile
n_x and l_x	number of mechanical connections on input and output ends in *x*-direction
n_y and n_w	number of mechanical connections on input end in *y* and *w* directions, respectively

Reaction force F_y is calculated from the equation:

$$F_y = F_{norm} \sin(\beta + \Delta\beta) = F_x \tan(\beta + \Delta\beta), \qquad (7)$$

where F_{norm} is the reaction force normal to cam profile at its contact point with the roller, F_x is the resultant force from output mechanical connections in x-direction (follower force), β is the pressure angle between force vectors F_{norm} and F_x without consideration of the cam center motion and $\Delta\beta$ is the correction of pressure angle due to the motion of cam center in x and *y-direction*s (refer to Figure 2) . Motion of the center of cam base circle requires the correction of the rotation angle of cam body by the value $\Delta\beta$ because the contact point between cam profile and the follower changes its position.

Resistance torque T_{Fx} for central cam is obtained from the following equation:

$$T_{Fx} = F_{norm} a_{norm} = \frac{F_x}{\cos(\beta + \Delta\beta)} \left(R_{base} + r_{roll} + x_{lift}\right) \sin\beta\,, \qquad (8)$$

where a_{norm} is the arm of the normal force F_{norm} , R_{base} is the radius of cam base circle, r_{roll} is the radius of roller and x_{lift} is cam follower lift (refer to Figure 2).

For eccentric cam resistance torque T_{Fx} is calculated as follows:

$$T_{Fx} = \frac{F_x}{\cos(\beta + \Delta\beta)} \left(\varepsilon + \sqrt{(R_{base} + r_{roll})^2 - \varepsilon^2} + x_{lift}\right) \sin\beta\,, \qquad (9)$$

where e is the follower eccentricity (offset).

As **Cam Profile** has no mass and moment of inertia, the governing equations of motion (Equation 6) are first-order differential equations. This implies that **Cam Profile** actually has not three but rather one and a half degree of freedom. Due to stiff connections cam body has to move closely together with the camshaft connected on input end (if any). In this case it is very important to specify the same initial conditions for **Cam Profile** and connected **Shaft** element. Moreover, same initial conditions of translation in x-direction have to be specified for the **Cam Profile** and connected output element (e.g. Plunger).

To prevent instability of motion at calculation start, initial angular velocity of **Cam Profile** has to coincide with the reference speed multiplied with transmission ratio (if defined). Reference speed is specified in **Calculation Control** dialog (refer to Section **26.2.6.3**.).

If **Cam Profile** is a boundary element (i.e. it has no input mechanical connections), then the center of cam base circle will not exhibit any motion. In this case the cam body will rotate around w axis with constant angular velocity equal to reference speed multiplied by transmission ratio.

As seen from the Equation 6, **Cam Profile** can have output connections only in x-direction. This means that cam body motion in y and w directions cannot be transferred to the elements connected on output end.

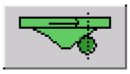
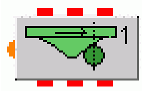
Damping and stiffness coefficients (functions) and preload forces/torques in Equation 6 are specified in the input dialogs of the respective mechanical connections (refer to Chapter 20).

Equation 6 is generally nonlinear if a **Shaft** element is attached to the input end of **Cam Profile** or output mechanical connection is variable. Variable connection implies that stiffness is a function of relative displacement and/or damping is a function of relative displacement or velocity.

If a **Shaft** element is attached to the input end of the **Cam Profile** and the angle between cam (local), mechanical connection (local) and shaft (global) coordinate systems is nonzero, BOOST Hydsim will transform coordinates and velocities of the connected elements into the local coordinate system of mechanical connection and calculate the resulting force. The components of this force will act on **Cam Profile** in local x and y directions.

If **Cam Profile** is a boundary element, the 2^{nd} and 3^{rd} equations of system (Equation 6) vanish. The 1^{st} equation reduces to the prescribed boundary function (acceleration/lift of the roller center of the reciprocating follower).

4.2. Cam Plate

Element Name:	**Cam Plate**	
Element Icon:		
Definition:	This element serves to define a cam plate of distributor-type injection pump (e.g. Bosch, Delphi) by acceleration of lift data.	
Connecting pins:	standard pins: 6 (all mechanical) special pins: 0 wire pins: 1	

☞ **Note:** Currently the Variable-step solver does not support this element.

☞ **Note:** On input side of **Cam Plate**, mechanical connections may be specified only in *w*-direction (rotation) while on output side only *x*-direction (translation) is allowed (refer to Figure 5).

Note: **Cam Plate** acceleration/lift data imply the axial acceleration/lift of the **Cam Plate** contact point with the reciprocating follower (plunger) in *x*-direction.

The mechanical model of **Cam Plate** with input-output connections is shown in Figure 5.

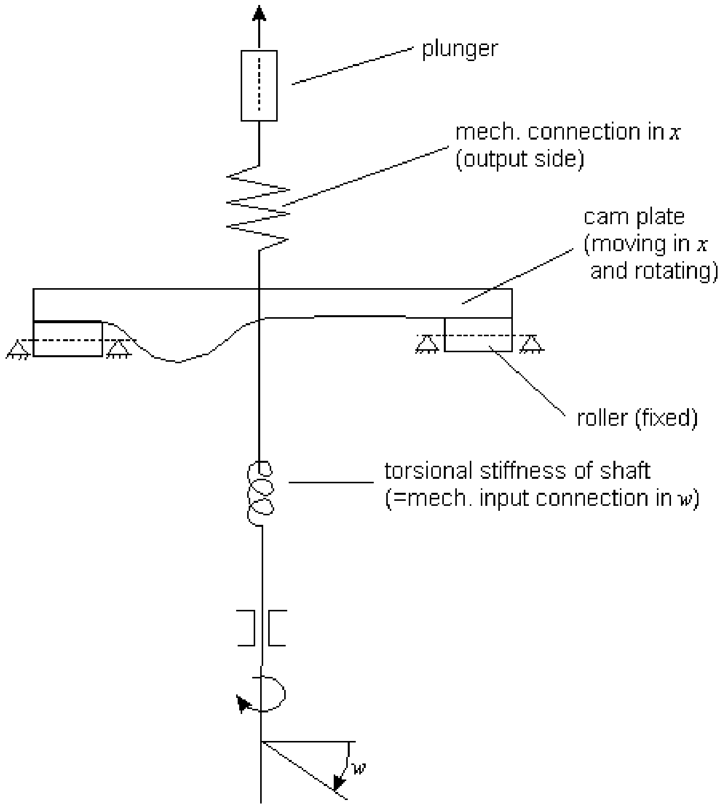


Figure 5: Mechanical Model of Cam Plate

4.2.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Transmission ratio vs. Reference speed	Specify transmission ratio between **Cam plate** rotation speed and Reference speed in **Calculation Control** dialog. Cam plate speed is then calculated as Reference speed multiplied with transmission ratio. **Note:** If an input connection in *w*-direction exists (from e.g.

	Shaft), then Transmission ratio input is disabled. In this case connected element (e.g. **Shaft**) speed is used as **Cam plate** speed.
Radius of cam plate	Specify the cam plate radius. The cam plate radius corresponds to R_{plate} in Figure 6.
Radius of roller	Specify the radius of roller. The roller radius corresponds to r_{roll} in Figure 6.
The radii of cam plate and roller are required only for the calculation of pressure angle, resulting forces on the cam profile, shaft torque, etc. For dynamic calculation alone they are not necessary.	
Plate lift at calculation start	Active only for **Cam Plate** profile defined in terms of the acceleration data. Specify the plate lift at calculation start. This will be used internally as constant of integration for initial plate velocity in x-direction.
Initial plate velocity at 1000 rpm	Specify initial plate velocity in x-direction at Cam Plate rotation with 1000 rpm, which will be used internally as constant of integration for calculation of initial plate lift or acceleration.
CAM PLATE PROFILE	
Shift angle relative to calculation start	Refer to shift angle relative to calculation start in Section 4.1.1.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in the 1st column will be multiplied with the scale factor as soon as the calculation is started.
Cam plate Lift in normal direction (x)	Click to activate cam plate lift in normal direction (*x*) in 2nd column.
Cam plate acceleration at 1000 rpm	Click to activate cam plate acceleration in normal direction (*x*) at 1000 rpm in 2nd column.
Cam profile table	Cam plate profile has to be specified for the entire calculation interval (End Reference angle) given in the **Calculation Control** dialog. However, if more than one revolution of the cam plate is considered in the calculation, it is sufficient to define the cam profile for one revolution of the camshaft (360 degrees). Intermediate points of cam plate profile are calculated by linear interpolation.

☞ **Note:** Follower lift as input data must be used with extreme caution. If the lift data are not precise enough, the numerical derivatives (velocity and acceleration) may contain coarse errors, and calculation results will be incorrect. The follower acceleration curve needs to be checked carefully within each calculation.

Intermediate points of the cam profile are calculated by linear interpolation if necessary. Cam plate profile has to be specified for the same interval of Reference angle as given in the **Calculation Control** dialog box. However, if more than one revolution of the cam is considered in the calculation, it is sufficient to define the cam profile for one revolution of the camshaft (360 degrees).

Cam Plate profile can be viewed in Cartesian co-ordinates by pressing the **View** bar. If pressed, an Impress Chart window is opened where the plate follower (plunger) acceleration/lift curve is plotted.

HERTZ STRESS CALCULATION	
Young's modulus of cam material	Specify the Young's modulus of cam material. It corresponds to E_{cam} in formula for Hertz stress calculation (refer to Equation 5).
Effective roller (contact) width	Specify the effective width of roller. It corresponds to b_{roll} in formula for Hertz stress calculation (refer to Equation 5).

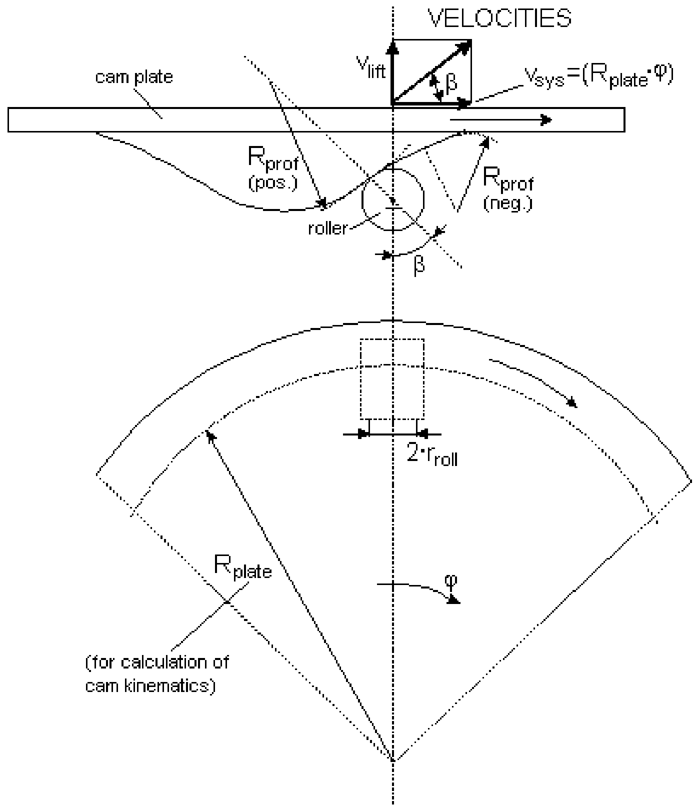


Figure 6: Geometry of Cam Plate

4.2.2. Initial Conditions

Path: **Element | Initial Conditions**

All initial values that are not specified in this dialog are set to 0. A description of initial conditions is given in Table 3.

Table 3: Initial Conditions of Cam Plate

w-angle	Specify initial angle of cam plate in rotation around axis *w*.
angular velocity in w-direction	Specify initial angular velocity of cam plate around rotation axis *w*.

☞ **Note:** Initial angular velocity of **Cam Plate** has to be specified as follows: if **Cam Plate** has no input connection in w-direction, then set initial velocity to Reference speed multiplied by transmission ratio. If **Cam Plate** has an input mechanical connection in w-direction (e.g. from **Shaft** element), then set initial velocity to Reference speed (defined in **Calculation Control**).

If nonzero initial conditions are specified, BOOST Hydsim may encounter convergence problems during the calculation of torsional distortion (if a **Shaft** element is connected on input end).

4.2.3. Modify Parameter

Cam Plate element has no modifiable parameters.

4.2.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box in front of output parameter must be checked.

Description of output parameters is given in Table 4.

Table 4: Output Parameters of Cam Plate

Lift of cam plate in x-direction	Lift of cam plate in *x*-direction is taken from lift table of the plate in **Input Data** dialog box or calculated by time integration of plate acceleration data.
velocity in x-direction of output end	Velocity of cam plate in *x*-direction is calculated from the angular velocity in *w*-direction $\dot{\varphi}_{plate}$ (refer to Figure 6): $$v_x = R_{plate}\dot{\varphi}_{plate} \tan\beta \text{ ,} \quad (10)$$ where β is the pressure angle of cam plate profile.
Rotation angle of cam plate about w axis	*w*-coordinate (rotation angle) of cam plate body. If **Cam Plate** is a boundary element (i.e. an input mechanical connection in *w*-direction does not exist), rotation angle of cam plate is the product of Reference speed $\dot{\varphi}_{ref}$, transmission ratio i_{tr} and time interval from the beginning of calculation t: $$w = \dot{\varphi}_{plate}\, t = i_{tr}\dot{\varphi}_{ref}\, t \text{ ,} \quad (11)$$ If an input mechanical connection in *w*-direction exists, then rotation angle is calculated from the equations governing the dynamics of the system.
angular velocity of cam plate rotation	Angular velocity in *w*-direction of cam plate body. If **Cam Plate** is a boundary element, its angular velocity is equal to Reference speed multiplied by transmission ratio. Otherwise (input mechanical connection in *w*-direction exists), angular velocity of cam plate is calculated at each step from the equations of motion of the system.

plate velocity in x-direction at 1000 rpm	Cam plate velocity in x-direction at 1000 rpm is obtained by numerical differentiation of lift or integration of acceleration data defined in **Input Data** dialog box.
plate acceleration in x-direction at 1000 rpm	Cam plate acceleration in *x*-direction at 1000 rpm is taken directly from the plate acceleration table in **Input Data** dialog box or derived by differentiating plate velocity at 1000 rpm.
pressure angle of cam plate profile	Pressure angle of cam plate profile corresponds to angle β in Figure 6. It is the angle between the force normal to cam profile and plate velocity in *x*-direction.
torsional distortion angle	Torsional distortion angle of cam plate is the difference between actual rotation angle and Reference angle which is calculated from the Reference speed. It is nonzero only if an input mechanical connection in w-direction exists (i.e. **Cam Plate** is not a boundary element).
reaction force normal to cam plate profile	Resistance force normal to cam profile F_{norm}. it is calculated from the reaction force of cam plate follower (plunger) in *x*-direction: $$F_{norm} = \frac{F_x}{cos\,\beta}, \qquad (12)$$ where F_x is the reaction force and β is the pressure angle.
torque of reaction force about w axis	Resistance torque about rotation axis *w* (induced by reaction force F_x). It is calculated from: $$T_{cp} = F_x R_{plate} \tan\beta \qquad (13)$$
torsional vibration velocity	Angular velocity of cam plate vibration relative to stationary rotation. Torsional vibration velocity is the difference between actual angular velocity in *w*-direction and reference speed, which is defined in **Calculation Control** dialog. It is nonzero only if an input mechanical connection in w-direction exists.
radius of curvature of cam plate profile	Radius of curvature of cam profile corresponds to radius R_{prof} (refer to Figure 6).
Hertz stress at cam-roller contact	Hertz stress at cam-roller contact is calculated from the formula: $$p_{Hertz} = 0.418\sqrt{\frac{E_{cam}F_{norm}}{b_{roll}}\left(\frac{1}{R_{prof}}+\frac{1}{r_{roll}}\right)} \qquad (14)$$
maximum Hertz stress till actual point	Maximum Hertz stress value from calculation start till actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).

4.2.5. Additional Information

- On input end, **Cam Plate** can be attached via mechanical connection to a **Shaft** element (Figure 5). In this case, the cam plate motion and shaft motion is considered separately. Mechanical connection between cam plate and shaft is a torsional spring and damper.
- To denote a rigid (or almost rigid) connection between the cam and shaft, the respective connection must possess very high stiffness (e.g. comparable to the shaft bearing stiffness).
- **Cam Plate** has one degree of freedom as it can be seen in Figure 5. Rotation of **Cam Plate** is coupled with its axial motion in x-direction. Equations governing the cam plate motion are discussed in Chapter 6.
- **Cam Plate** has no mass and moment of inertia. Its moment of inertia can be added to the moment of inertia of the connected **Shaft** element.
- If **Cam Plate** does not have an input mechanical connection in w-direction, its angular velocity in is equal to reference speed multiplied by transmission ratio. **Reference speed** is defined in **Calculation Control** dialog box.

4.2.6. Equations of Motion

Motion of **Cam Plate** is governed by the following equations:

$$\begin{aligned}
&\sum_{j=1}^{l_x} c_{xj}(\dot{x}-\dot{x}_j)+\sum_{j=1}^{l_x} k_{xj}(x-x_j)=\sum_{j=1}^{l_x} F_{0xj}, \\
&\sum_{i=1}^{n_w} c_{wi}(\dot{w}-\dot{w}_i)+\sum_{i=1}^{n_w} k_{wi}(w-w_i)=\sum_{i=1}^{n_w} T_{0i}-T_{Fx},
\end{aligned} \tag{15}$$

where

x	coordinate of Cam Plate axial motion
w	rotation angle of Cam Plate
x_j	coordinate of the *j-th* connected element on output end
c_{xj} and k_{xj}	damping and stiffness coefficients (functions) of the *j-th* output mechanical connection in *x-direction*
c_{wi} and k_{wi}	torsional damping and stiffness coefficients (functions) of the *i-th* input mechanical connection in *w* direction
F_{0xj}	preload force of the *j-th* output mechanical connection in *x-direction*
T_{0i}	preload torque of the *i-th* input mechanical connection in *w* direction
T_{Fx}	resistance torque caused by the resultant force from output mechanical connections
l_x	number of mechanical connections on output end in *x-direction*
n_w	number of mechanical connections on input end in *w* direction.

Normally l_x=1 and n_w=1. As Cam Plate has no mass and moment of inertia, the governing equations of motion (equation 15) are first-order differential equations. Both equations are coupled since axial Cam Plate motion is uniquely expressed through plate rotation:

$$x = wR_{plate} \tan\beta , \tag{16}$$

where R_{plate} is the radius of **Cam Plate** and β is the pressure angle between *x*/*w* axes, and the normal line to cam profile at its contact point with the roller (refer to Figure 6).

Hence **Cam Plate** has only one degree of freedom (more precisely, half a degree of freedom). Due to stiff connection **Cam Plate** has to rotate closely together with the Shaft or torsional Boundary element connected on input end (if any). Furthermore, in axial direction it has to move closely together with the element connected on output end (e.g. **Plunger**). Thus, it is very important to specify the same initial conditions of rotation in *w* direction for **Cam Plate** and connected **Shaft** as well as the same initial conditions of translation in *x-direction* for the **Cam Plate** and connected output element (**Plunger**).

To prevent instability of rotation at calculation start, initial angular velocity of **Cam Plate** has to coincide with the reference speed multiplied with transmission ratio (if defined). Reference speed is specified in **Calculation Control** dialog (refer to Section **26.2.6.3**).

If **Cam Plate** is a boundary element (i.e. it has no input mechanical connections), it will rotate around *w* axis with constant angular velocity equal to reference speed multiplied by transmission ratio.

Resistance torque T_{Fx} is calculated from the equation:

$$T_{Fx} = F_x R_{plate} \tan\beta \,, \qquad (17)$$

where F_x is the resultant force from output mechanical connections in *x-direction* (plunger reaction force).

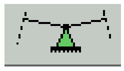
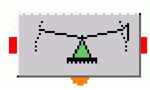
As seen from Equation 15, **Cam Plate** can have input connections only in *w* direction and output connections only in *x-direction*. Motion in both directions is coupled.

Damping and stiffness coefficients (functions) and preload forces/torques in Equation 15 are specified in the input dialogs of respective mechanical connections (refer to Chapter 20).

Equation 15 is nonlinear if a **Shaft** element is attached to the input end of **Cam Plate** or one of the mechanical connections is variable. However, if **Cam Plate** is a boundary element, Equation 15 reduces to the prescribed boundary function (acceleration/lift).

5. LEVER

5.1. Rocker Arm (Pushrod)

Element Name:	**Rocker Arm (Pushrod)**	
Element Icon:		
Definition:	This element serves to define the properties of pushrod-actuated Rocker Arm.	
Connecting pins:	standard pins: 2 (mechanical) special pins: 0 electrical pins: 1 Only one input and one output mechanical connection can be specified.	

Note: Currently the Variable-step solver does not support this element.

A schematic of **Rocker Arm (Pushrod)** element is shown in Figure 7.

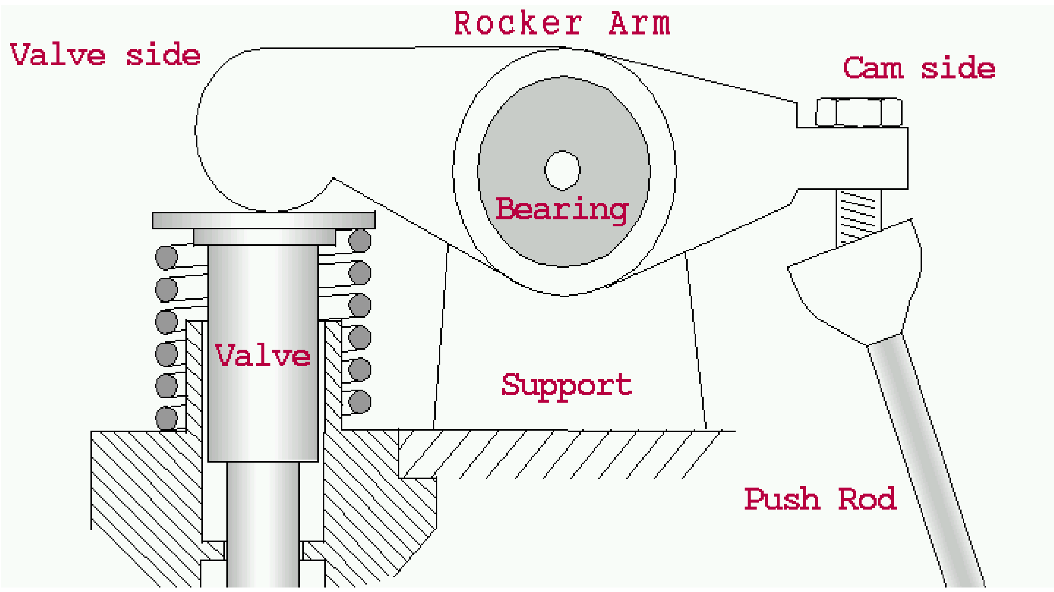


Figure 7: Rocker Arm (Pushrod) Schematic

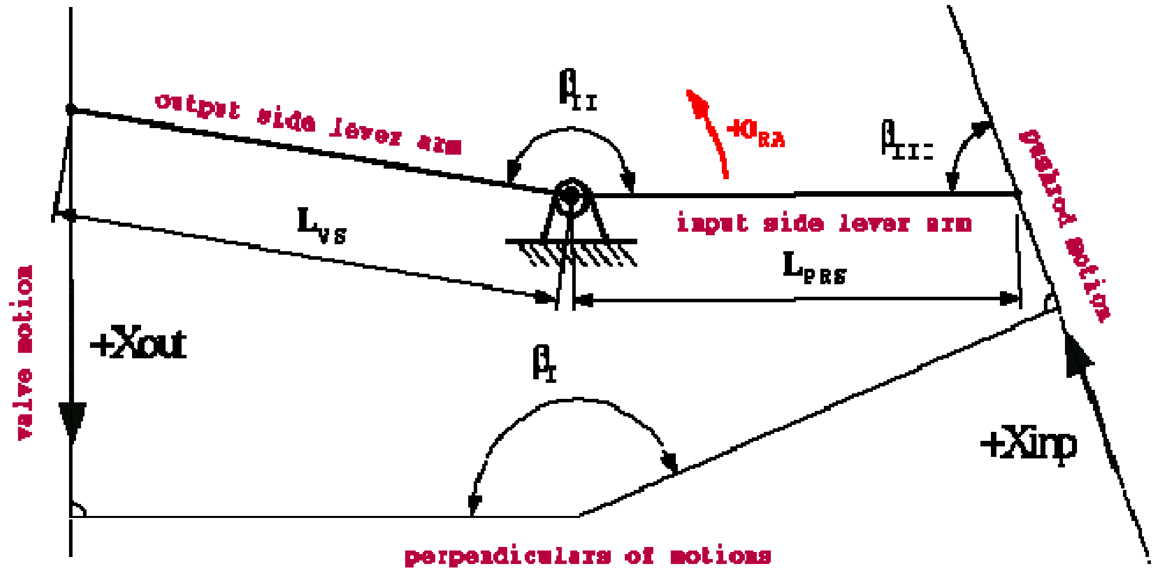
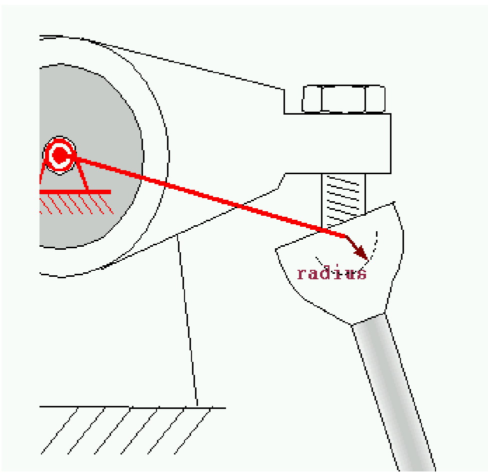
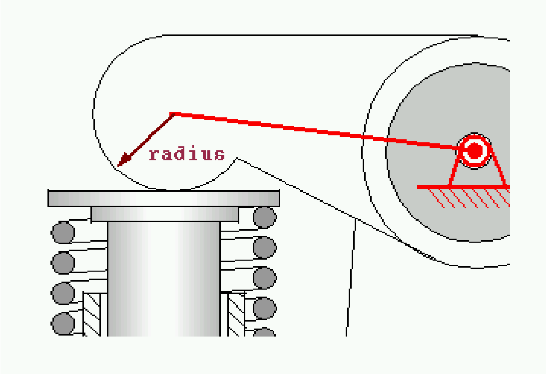
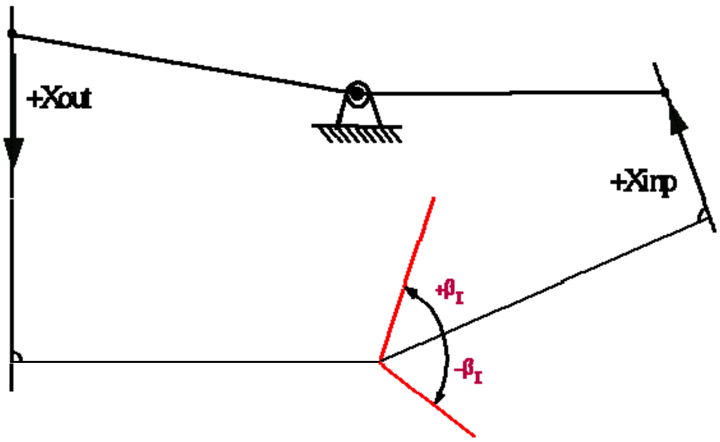
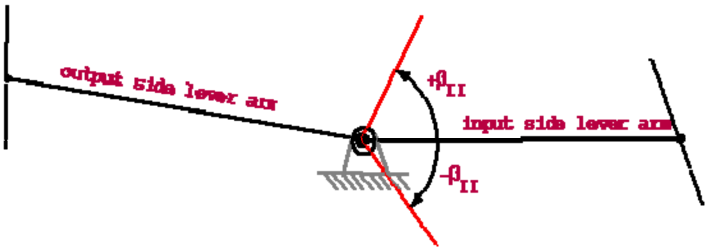


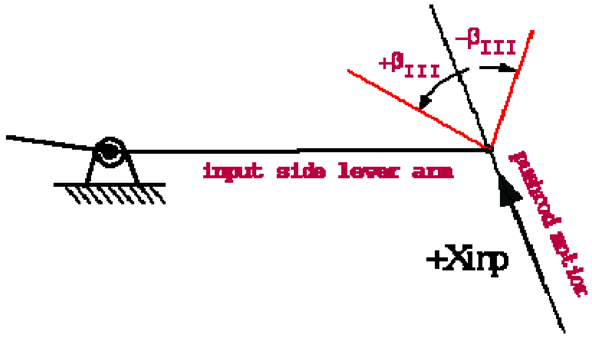
Figure 8: Rocker Arm (Pushrod) Geometry I

5.1.1. Input Parameters

Description of input data for the **Rocker Arm (Pushrod)**:

Element name	The name of the element is specified as default.
Moment of inertia about rotation axis	Specify moment of inertia of rocker arm with roller about rotation axis (I_{RA}).
Geometric data	
Length of input side lever arm (pushrod)	Specify length of input side lever arm (L_{PRS}). Input arm is the line between support and contact arc center on input side (refer to Figure 8).  **Figure 9: Input Side Lever Arm**

Length of output side lever arm (valve)	Specify length of output side lever arm (L_{VS}). Output arm is line between support and contact arc center on output side (refer to Figure 8).  **Figure 10: Output Side Lever Arm**
Angle between perpendiculars of motion	Specify angle between perpendiculars to positive input and negative output motion (β_I).  **Figure 11: Angle Between Perpendiculars of Motions**
Angle between input/output lever arms	Specify angle between input and output lever arms (β_{II}).  **Figure 12: Angle Between Input/Output Lever Arms**

Angle between pushrod motion/input arm	Specify angle between positive pushrod motion and input arm (β_{III}).  **Figure 13: Angle Between Pushrod Motion/Input Arm**
Elasticity of rocker arm	If check box is inactive (or stiffness is zero), elastic deformation of respective rocker arm will not be calculated.
Stiffness normal to input side arm (pushrod)	Specify stiffness normal to input side arm (pushrod) (k_{arm_inp}). This stiffness is used for the calculation of elastic deformation of input side arm. Elastic deformation is considered in correction of element input x-coordinate (X_{inp}) (refer to Figure 17).
Stiffness normal to output side arm (valve)	Specify stiffness normal to output side arm (valve) (k_{arm_out}). This stiffness is used for the calculation of elastic deformation of output side arm. Elastic deformation is considered in correction of element output x-coordinate (X_{out}) (refer to Figure 17).
Elastic rocker support	If check box is inactive (or rocker arm mass is zero), support motion will not be calculated.
Rocker arm mass	Specify mass of rocker arm with roller (m_{RA}) (refer to Figure 16).
Support bearing stiffness	Specify support bearing stiffness (k_{br}) (refer to Figure 16).
Support bearing damping	Specify support bearing damping (c_{br}) (refer to Figure 16).
Direction of Element motion	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
x(e1)	
y(e2)	
z(e3)	
Bearing Friction	Press this bar to specify the bearing friction.
Friction parameters	If check box is inactive (or bearing diameter is zero), friction data input is disabled and friction torque will not be calculated.
Bearing (shaft) diameter	Specify bearing (shaft) diameter (R_{br}) (refer to Figure 15).

Bearing friction force	Specify bearing friction force (F_{frict_0}) (refer to Figure 15).
Coulomb friction coefficient	Specify friction coefficient (μ_{br}) (refer to Figure 15).
Viscous friction coefficient	Specify viscous friction coefficient (λ_{br}) (refer to Figure 15).

5.1.2. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial conditions:

Angle	Specify the initial angle of rocker arm around rotation axis.
Angular Velocity	Specify the initial angular velocity of rocker arm around rotation axis.

5.1.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

Moment of inertia about rotation axis	SI Unit:	kgm^2
Stiffness normal to input side arm (pushrod)	SI Unit:	N/m
Stiffness normal to output side arm (valve)	SI Unit:	N/m
Stiffness of rocker arm support bearing	SI Unit:	N/m
Damping of rocker arm support bearing	SI Unit:	Ns/m
Friction force in rocker arm bearing	SI Unit:	N
Coulomb friction coefficient in bearing	SI Unit:	-
Viscous friction coefficient in bearing	SI Unit:	Ns/m

5.1.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the option box in front of output parameter must be activated.

Description of output parameters:

lift of input side arm (pushrod) along pushrod motion	Input x-coordinate (X_{inp}) of input side arm end (refer to Figure 8).

lift of input side arm (pushrod) due to elastic deformation	Deformation (ε_{inp}) is calculated according to the Equations 39 and 41. It is nonzero only if check box “Elasticity of rocker arms” is active.
velocity of input side arm (pushrod) along pushrod motion	Velocity in input x-direction of input side arm end.
lift of output side arm (valve) along follower motion	Output x-coordinate (X_{out}) of output side arm end (refer to Figure 8).
lift of output side arm (valve) due to elastic deformation	Deformation (ε_{out}) is calculated according to the Equations 40 and 42. It is nonzero only if check box “Elasticity of rocker arms” is active.
velocity of output side arm (valve) along follower motion	Velocity in output x-direction of output side arm end.
rotation angle of rocker arm	Angle of rocker arm around rotation axis (α_{RA}).
angular velocity around rotation axis	Angular velocity of rocker arm around rotation axis ($\dot{\alpha}_{RA}$).
angular acceleration around rotation axis	Angular acceleration of rocker arm around rotation axis (refer to Equation 26).
support lift along input side arm pushrod motion	Support lift (x_{su_inp}) is calculated according to the Equation 37. It is nonzero only if check box “Elastic rocker support” is active.
support lift along output side arm follower motion	Support lift (x_{su_out}) is calculated according to the Equation 38. It is nonzero only if check box “Elastic rocker support” is active.
amplitude of support motion (lift)	Amplitude of support motion is calculated from coordinates of support in local *x* and *y*-direction (refer to Equation 36).
friction force in support bearing	Friction force in support bearing is calculated from the formula: $F_{frict} = F_{frict\_0} + \mu_{br} F_{su} + \lambda_{br} \frac{\omega_{RA}}{R_{br}}$ (18) It is nonzero only if check box “Friction parameters” is active.

friction torque in support bearing	Friction torque in support bearing is calculated from the formula: $$T_{frict} = sign(\omega_{RA}) F_{frict} R_{br} \qquad (19)$$ It is nonzero only if check box “Friction parameters” is active.
reaction force on input arm (pushrod) along pushrod motion	Reaction force on input arm along pushrod motion (F_{Cinp}) is result of preload, stiffness and damping forces of input mechanical connection.
reaction force on output arm (valve) along follower motion	Reaction force on output arm along follower motion (F_{Cout}) is result of preload, stiffness and damping force of output mechanical connection.
resultant torque around rotation axis	Resultant torque around rotation axis is calculated according to the formula: $$T_{res} = T_{inp} - T_{out} - T_{frict}$$ (refer to Equations 23, 24 and 25)
maximum torque around rotation axis	Maximum torque value from calculation start untill actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).

5.1.5. Equations

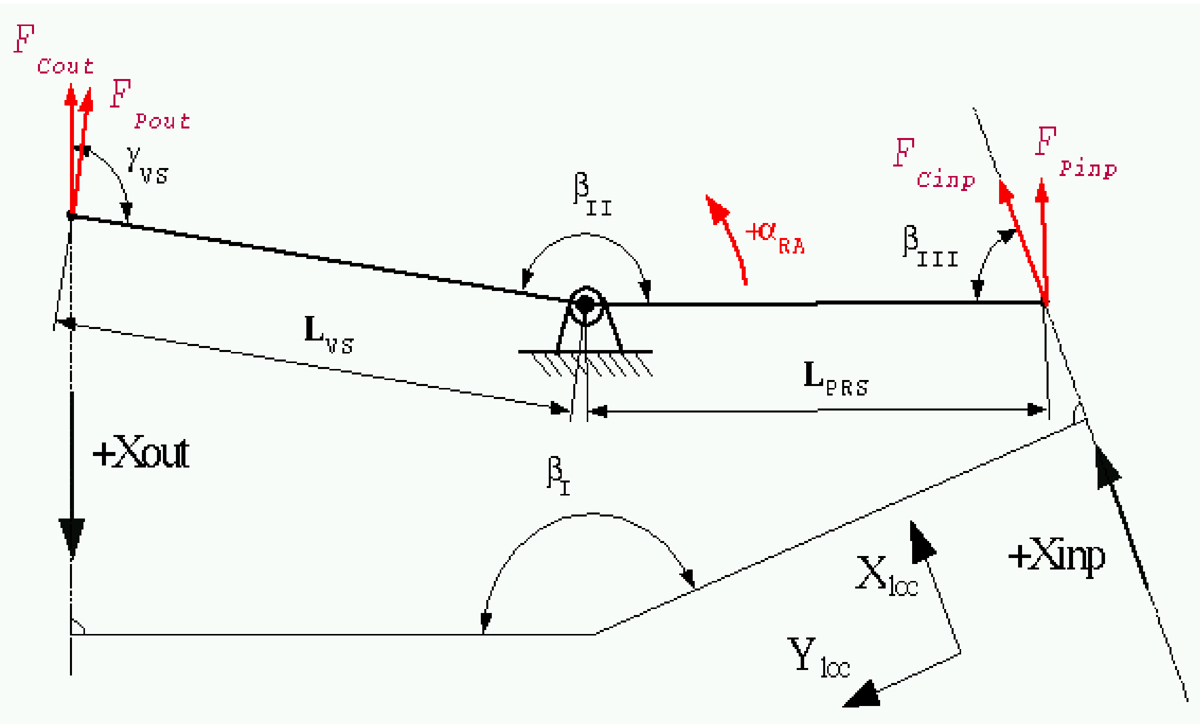


Figure 14: Rocker Arm (Pushrod) Geometry II

Angle between output arm and its follower motion:

$$\gamma_{VS} = \pi + \beta_I - \beta_{II} - \beta_{III} \tag{20}$$

Perpendicular components of connection forces on input and output side:

$$F_{Pinp} = F_{Cinp} \sin(\beta_{III} + \alpha_{RA}) \tag{21}$$

$$F_{Pout} = F_{Cout} \sin(\gamma_{VS} - \alpha_{RA}) \tag{22}$$

Torque on rocker arm (as result of input and output connection forces):

$$T_{inp} = F_{Pinp} L_{PRS} \tag{23}$$

$$T_{out} = F_{Pout} L_{VS} \tag{24}$$

Torque on rocker arm (as result of bearing friction):

$$T_{frict} = sign(\omega_{RA}) \left(F_{frict\_0} + \mu_{br} F_{su} + \lambda_{br} \frac{\omega_{RA}}{R_{br}} \right) R_{br} \tag{25}$$

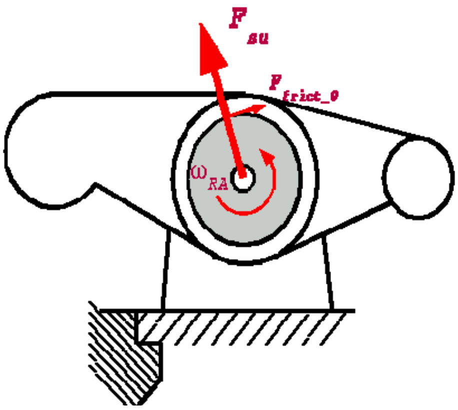


Figure 15: Support Bearing Friction

Angular acceleration around rotation axis of rocker arm:

$$\ddot{\alpha}_{RA} = \frac{T_{inp} - T_{out} - T_{frict}}{I_{RA}} \tag{26}$$

Forces on rocker arm support (as result of input and output connection forces) in local *X*-*Y* coordinate system:

$$F_{Xsu} = F_{Cinp} + F_{Cout} \cos(\pi - \beta_I) \tag{27}$$

$$F_{Ysu} = -F_{Cout} \sin(\pi - \beta_I) \tag{28}$$

Resultant force on rocker arm support (as result of input and output connection forces):

$$F_{su} = \sqrt{F_{Xsu}^2 + F_{Ysu}^2} \tag{29}$$

Forces on rocker arm support (as result of stiffness and damping of bearing support) in local *XY* coordinate system:

$$F_{Xbr\_c} = c_{br} \dot{x}_{su} \tag{30}$$

$$F_{Ybr\_c} = c_{br} \dot{y}_{su} \tag{31}$$

$$F_{Xbr\_k} = k_{br} x_{su} \tag{32}$$

$$F_{Ybr\_k} = k_{br} y_{su} \tag{33}$$

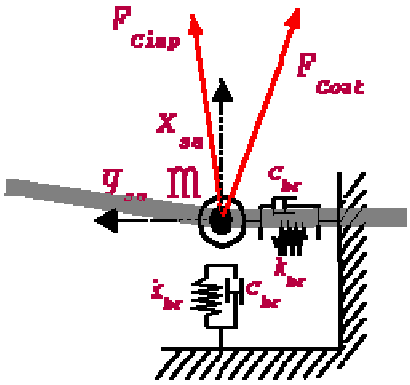


Figure 16: Elastic Rocker Support

Acceleration in local x and y direction of support (rotation axis of rocker arm):

$$\ddot{x}_{su} = \frac{F_{Xsu} - F_{Xbr\_c} - F_{Xbr\_k}}{m} \tag{34}$$

$$\ddot{y}_{su} = \frac{F_{Ysu} - F_{Ybr\_c} - F_{Ybr\_k}}{m} \tag{35}$$

Amplitude of support motion:

$$A_{su} = \sqrt{x_{su}^2 + y_{su}^2} \tag{36}$$

Lift of input and output side arms due to support motion:

$$x_{su\_inp} = A_{su} \cos(y_{bd}) \tag{37}$$

$$x_{su\_out} = -A_{su} \cos(\gamma_{bd}) \cos(\pi - \beta_I) \tag{38}$$

Elastic deformation of input and output side arm ends:

$T_{inp} > T_{out}$

$$\varepsilon_{inp} = F_{Pout} \frac{L_{VS}}{L_{PRS}} k_{arm\_inp} \tag{39}$$

$$\varepsilon_{out} = F_{Pout} k_{arm\_out} \tag{40}$$

$T_{inp} \le T_{out}$

$$\varepsilon_{inp} = F_{Pinp} k_{arm\_inp} \tag{41}$$

$$\varepsilon_{out} = F_{Pinp} \frac{L_{PRS}}{L_{VS}} k_{arm\_out} \tag{42}$$

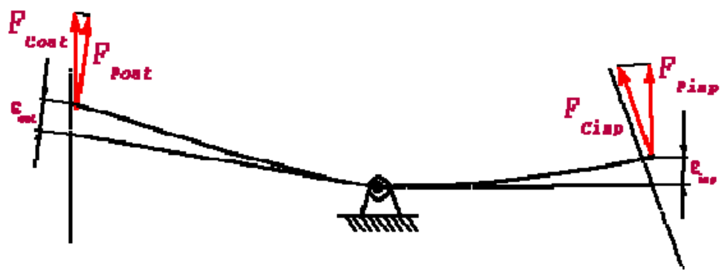


Figure 17: Elasticity of Rocker Arm

Lift of input and output arms due to elastic deformations of rocker arm:

$$x_{df\_inp} = \varepsilon_{inp} \cos\left(\beta_{III} + \alpha_{RA} - \frac{\pi}{2} \right) \tag{43}$$

$$x_{df\_out} = -\varepsilon_{out} \cos\left(\gamma_{VS} - \alpha_{RA} - \frac{\pi}{2} \right) \tag{44}$$

Lift of input and output arms due to pure rotation of rocker arm:

$$x_{RA\_inp} = L_{PRS} \left(\cos(\beta_{III}) - \cos(\beta_{III} + \alpha_{RA}) \right) \tag{45}$$

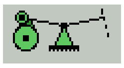
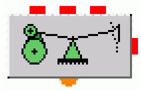
$$x_{RA\_out} = L_{VS} \left(\cos(\gamma_{VS} - \alpha_{RA}) - \cos(\gamma_{VS}) \right) \tag{46}$$

Resultant lift of input and output arms:

$$x_{inp} = x_{RA\_inp} + x_{df\_inp} + x_{su\_inp} \tag{47}$$

$$x_{out} = x_{RA\_out} - x_{df\_out} + x_{su\_out} \tag{48}$$

5.2. Rocker Arm (Cam)

Element Name:	**Rocker Arm (Cam)**
Element Icon:	
Definition:	This element serves to define the properties of cam-actuated Rocker Arm.
Connecting pins:	standard pins: 4 (mechanical) special pins: 0 electrical pins: 1 Three input mechanical connections in *x*, *y* and *w*-directions can be specified. Only one output mechanical connection in *x*-direction can be specified. 

Note: Currently the Variable-step solver does not support this element.

The schematic of **Rocker Arm (Cam)** is shown in Figure 18.

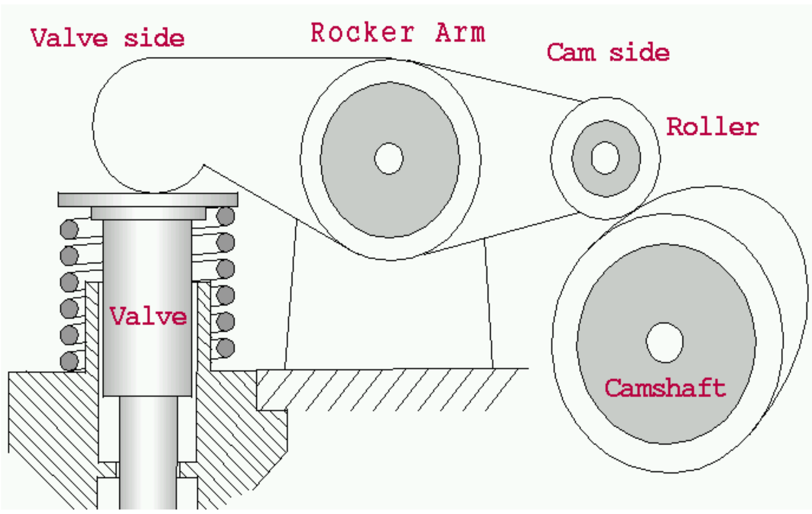


Figure 18: Rocker Arm (Cam) Schematic

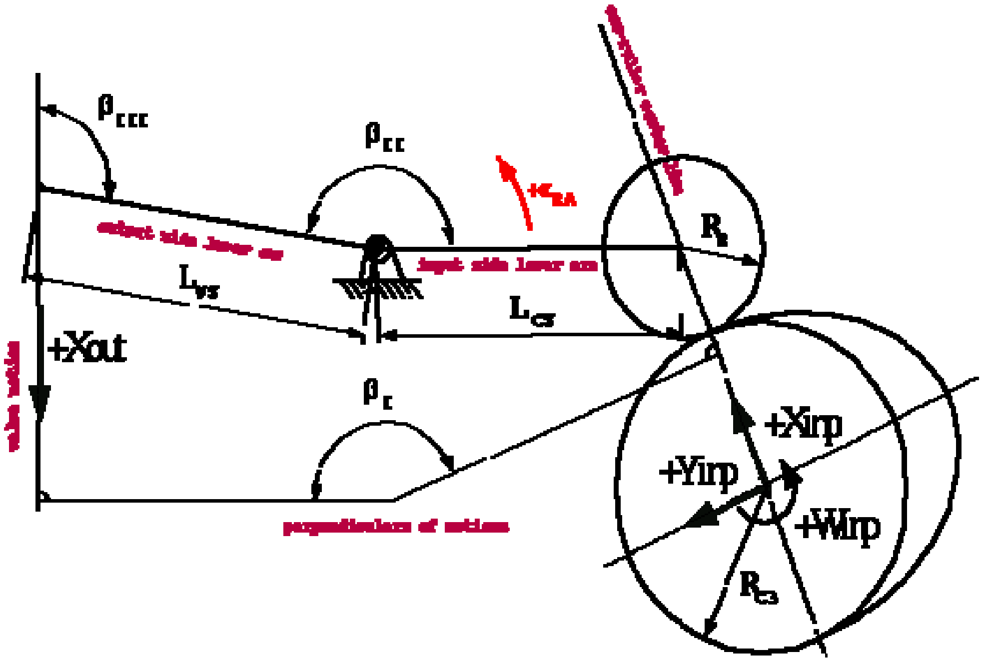
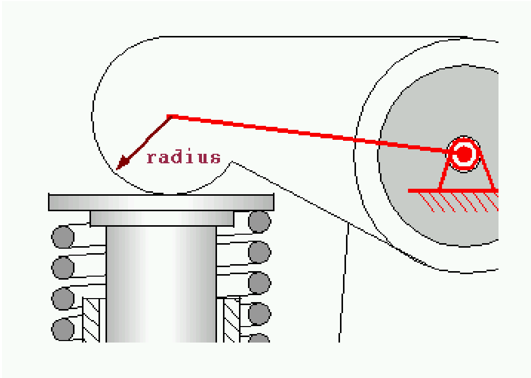
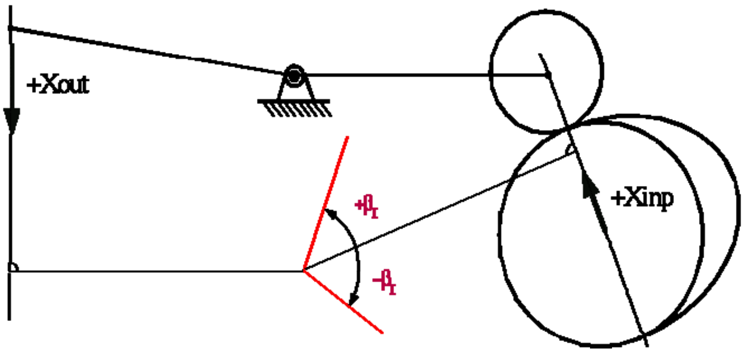
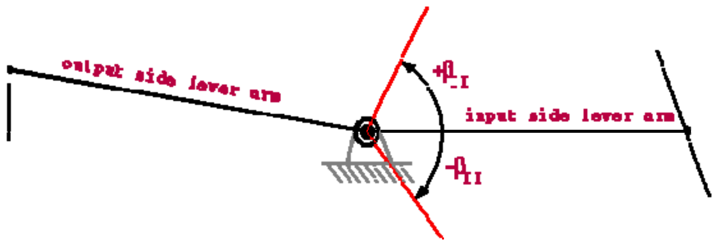


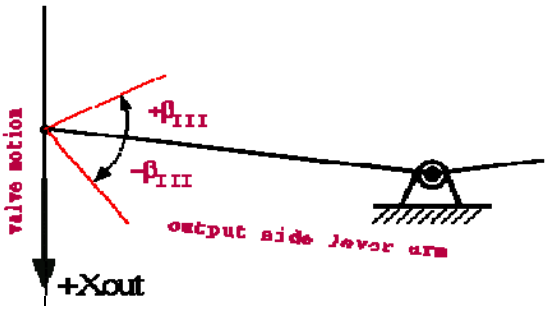
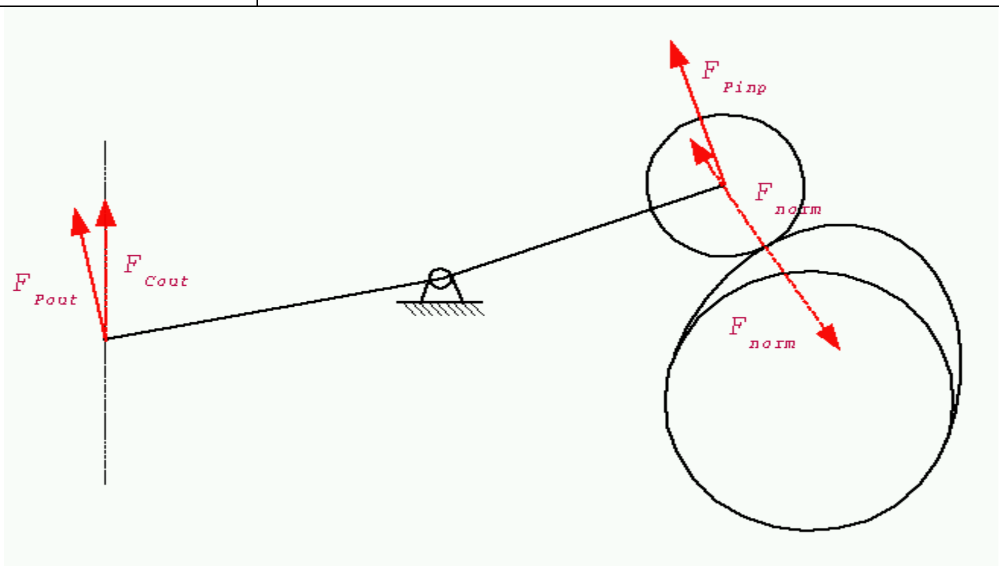
Figure 19: Rocker Arm (Cam) Geometry I

5.2.1. Input Parameters

Description of input data for the **Rocker Arm (Cam)**:

Element name	The name of the element is specified as default.
Cam Profile	Press this bar to specify the cam profile in polar coordinates and characteristics of contact between roller and cam body (refer to Section 5.2.2).
Rocker Bearing Friction	Press this bar to specify the friction parameter inside bearing. (refer to Section 5.2.3).
Inertia data	
Rocker arm mass	Specify mass of rocker arm with roller (m_{RA}).
Moment of inertia about rotation axis	Specify moment of inertia of rocker arm with roller about rotation axis of rocker arm (I_{RA}).
Geometric data	
Length of input side lever arm (cam)	Specify length of input side lever arm (L_{CS}). Input arm is line between support and center of roller rotation (refer to Figure 19).

Length of output side lever arm (valve)	Specify length of output side lever arm (L_{VS}). Output arm is line between support and contact arc center on output side (refer to Figure 19).  **Figure 20: Output Side Lever Arm**
Angle between perpendiculars of motion	Specify angle between perpendiculars on initial positive input motion and positive output motion (β_I).  **Figure 21: Angle Between Perpendiculars of Motions**
Angle between input/output lever arms	Specify angle between input and output lever arms (β_{II}).  **Figure 22: Angle between input/output lever arms**

Angle between output arm/valve motion	Specify angle between output arm and negative valve motion (β_{III}).  **Figure 23: Angle Between Output Arm/Valve Motion**
Elasticity of lever arms	If check box is inactive (or stiffness is zero), elastic deformation of respective lever arm will not be calculated.
Stiffness normal to input side arm (cam)	Specify stiffness normal to input side arm (k_{arm_inp}). This stiffness is applied for the calculation of elastic deformation of input side arm. This deformation is used for correction of roller position in contact with cam profile (refer to Figure 24 and Figure 25).
Stiffness normal to output side arm (valve)	Specify stiffness normal to output side arm (k_{arm_out}). This stiffness is applied for the calculation of elastic deformation of output side arm. Calculated deformation is used in correction of element output x-coordinate (X_{out}) (refer to Figure 25).
 **Figure 24: Forces On Rocker Arm**	

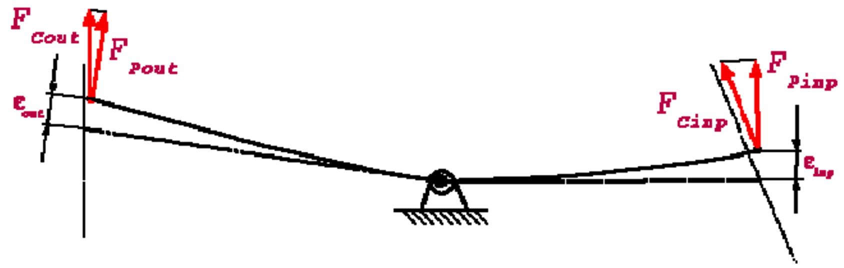


Figure 25: Elasticity of Lever Arms

Elastic rocker support	If check box is inactive (or rocker arm mass is zero), support motion will not be calculated.
Support bearing stiffness	Specify support bearing stiffness (k_{br}) (refer to Figure 26).
Support bearing damping	Specify support bearing damping (c_{br}) (refer to Figure 26).

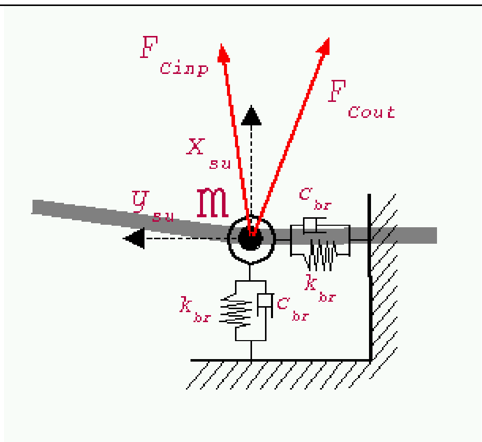


Figure 26: Elastic Rocker Support

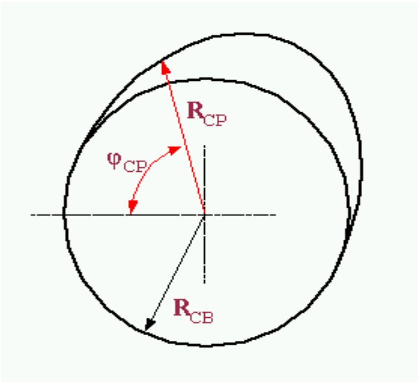
5.2.2. Rocker Arm - Cam Profile

This **Input Data Dialog Box** of **Cam Profile** serves to define the Cam Profile in polar co-ordinates and characteristic of contact between roller and cam profile.

Description of input data:

Cam body rotation	
clockwise	For Rocker Arm (cam actuated) model as it is presented on Figure 19; this is negative cam body rotation.
counterclockwise	For Rocker Arm (cam actuated) model as it is presented on Figure 19; this is positive cam body rotation.
Transmission ratio vs. Reference speed	Specify transmission ratio between cam rotation speed and Reference speed in **Calculation Control** dialog. Cam speed is calculated as Reference speed multiplied with transmission ratio. **Note:** If an input connection in *w*-direction exists (from e.g.

	Shaft), then Transmission ratio input is disabled. In this case speed of connected element (**Shaft**) speed is used as **Cam** speed.
Radius of cam base circle	Specify radius of cam base circle (R_{CB}) (refer to Figure 19 and Figure 27).
Radius of roller	Specify radius of roller (R_R) (refer to Figure 19).

Cam Profile (polar coordinates)	
Shift angle relative to calculation start	Specify the shift angle of cam profile relative to the start position. Positive angle implies that from the calculation start up to the shaft rotation angle, less than specified shift angle, the roller rests on the cam base circle. Hence, if positive shift angle is defined, the initial conditions of the rocker arm must be equal to zero. Negative shift angle implies the calculation start from the cam profile value corresponding to the shift angle value in the cam profile table. In this case, the appropriate number of rows in the table will be skipped until the row with the cam angle equal to shift angle is found. Clearly, initial conditions of the rocker arm motion for the negative shift angle must be nonzero. Shift angle must be defined in the range 0…360 degrees. **Note:** Negative shift angle is not implemented in BOOST Hydsim.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in the 1st column will be multiplied with the scale factor as soon as the calculation is started.
Cam Profile table	Specify angles (φ_{CP}) in ascending order in first column. Specify polar coordinate (R_{CP}) of cam profile in second column.  **Figure 27: Polar Coordinate of Cam Profile** Polar coordinate of **cam profile** is calculated according to the actual angle position of the cam body and position of roller on it.
Contact stiffness (Hertz stress caclulation)	
Young's modulus of cam material	Specify Young's modulus of cam material (E_{cam}).
Poisson's ratio of cam material	Specify Poisson's ratio of cam material (υ_{cam}).

Effective roller (contact) width	Specify effective roller width (b_R).
Contact damping ratio	Specify contact damping ratio (r_C).

Direction of Cam X-axis in global coordinates	
x (e_1) **y (e_2)** **z (e_3)**	Specify unit vector of cam x-axis in the global coordinate system: $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in shaft (global) coordinate system, e_1, e_2, e_3 are the components of cam x-axis unit vector in global *x*, *y* and *z* direction (see Figure 28).

☞

Note: Input "Cam X-axis unit vector in global coordinates" is active only if a **Shaft** element is connected to **Rocker Arm (Cam)** via an input mechanical connection.

Note: BOOST Hydsim mechanical part is based on theory of vibration of 2D multi-body systems (translation in *x* and *y* and rotation around *w*). Therefore 3rd translational coordinate *z* is inactive (grayed out) in the input dialog.

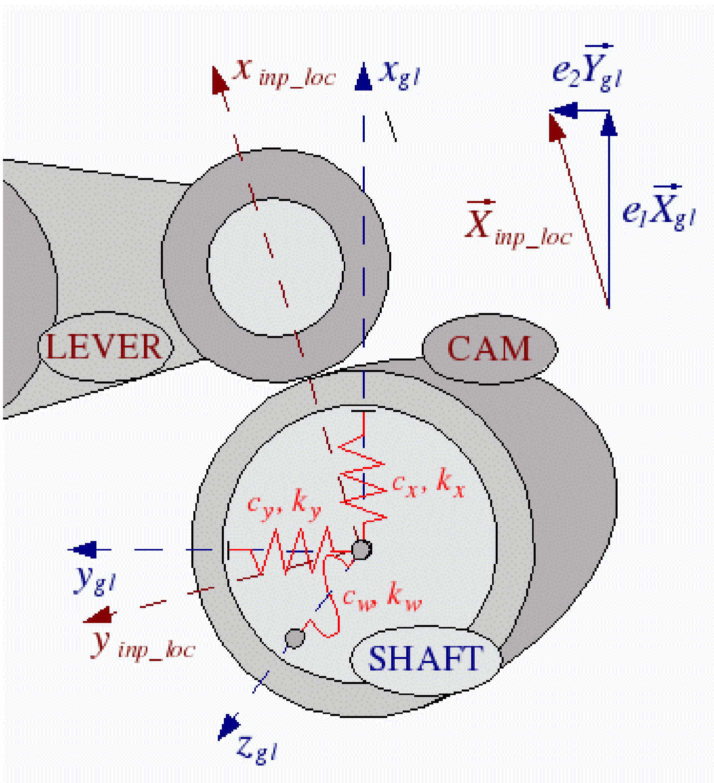


Figure 28: Cam Profile with connections to Shaft

5.2.3. Bearing Friction

This **Input Data Dialog Box** of **Bearing Friction** serves to define the characteristics of friction inside rocker arm bearing.

Description of input data:

Friction parameters	If check box is inactive (or bearing diameter is zero), friction data input is disabled and friction torque will not be calculated.
Bearing (shaft) diameter	Specify bearing (shaft) diameter (R_{br}) (refer to Figure 29).
Bearing friction force	Specify bearing friction force (F_{frict_0}) (refer to Figure 29).
Coulomb friction coefficient	Specify friction coefficient (μ_{br}) (refer to Figure 29).
Viscous friction coefficient	Specify viscous friction coefficient (λ_{br}) (refer to Figure 29).

Friction torque is calculated according to the formula:

$$T_{frict} = sign(\omega_{RA})\left(F_{frict\_0} + \mu_{br} F_{su} + \lambda_{br} \frac{\omega_{RA}}{R_{br}}\right) R_{br}$$

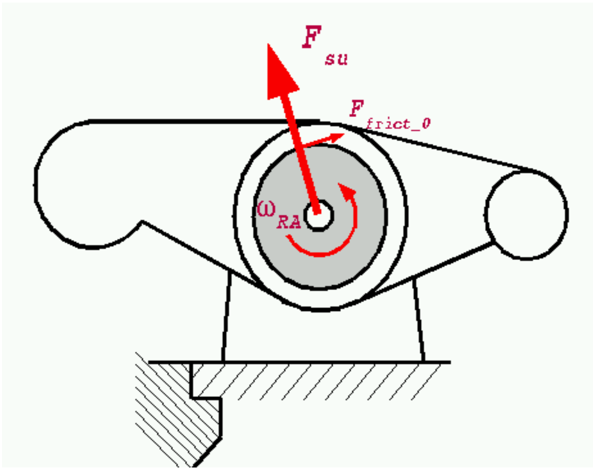


Figure 29: Support bearing friction

5.2.4. Initial Conditions

Path: **Element** | **Initial Conditions**

Description of initial conditions:

Cam Body (input)	
Initial translation	
x-coordinate	Specify the initial position of cam center in *x*-direction.
velocity in x-direction	Specify the initial velocity of cam center in *x*-direction.
y-coordinate	Specify the initial position of cam center in *y*-direction.

velocity in y-direction	Specify the initial velocity of cam center in *y*-direction.
Initial rotation	
w-angle	Specify the initial angle of cam base body in *w*-direction.
angular velocity in w-direction	Specify the initial angular velocity of cam base body rotation in *w*-direction.
Valve Side (output)	

Initial translation	
x-coordinate	Specify the initial position of output side arm end in output *x*-direction
velocity in x-direction	Specify the initial velocity of output side arm end in output *x*-direction.

5.2.5. Modifiable Parameter

Path: **Element│ Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is checked, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

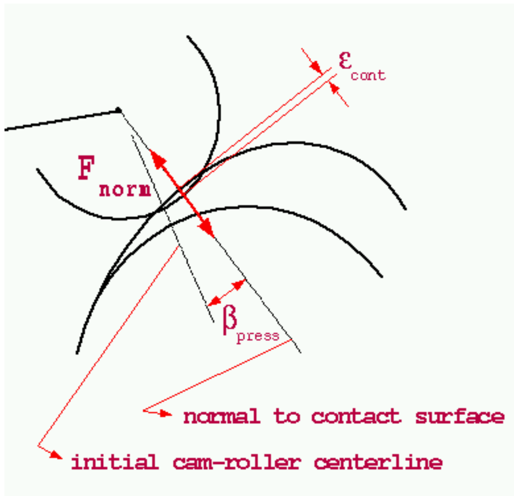
moment of inertia about rotation axis	SI Unit:	kgm^2
stiffness normal to input side arm (cam)	SI Unit:	N/m
stiffness normal to output side arm (valve)	SI Unit:	N/m
stiffness of rocker arm support bearing	SI Unit:	N/m
damping of rocker arm support bearing	SI Unit:	Ns/m
friction force in rocker arm bearing	SI Unit:	N
Coulomb friction coefficient in bearing	SI Unit:	-
viscous friction coefficient in bearing	SI Unit:	Ns/m

5.2.6. Output Parameters

Path: **Element│ Store Results**

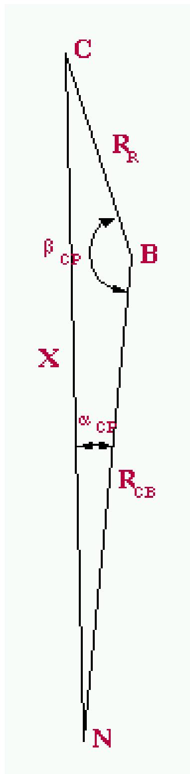
To activate output parameter, the option button in front of output parameter must be checked.

Description of output parameters:

cam profile (contact point) lift along cam-roller center axis	This is a projection of the actual polar coordinate of cam profile (distance between contact point and center of cam body rotation) to cam-roller center axis.
elastic deformation at contact point (normal to cam profile)	Elastic deformation at contact point is calculated from the position of rocker arm support, angle of rocker arm, angle of cam body and position of cam body center. This elastic deformation is used for the calculation of the contact force.  **Figure 30: Elastic Deformation at Contact Point**
elastic deformation of cam side arm (normal to cam profile)	Deformation (ε_{inp}) is calculated according to the formula: for $T_{inp} > T_{out}$: $\varepsilon_{inp} = F_{Pout} \dfrac{L_{VS}}{L_{CS}} / k_{arm\_inp}$ for $T_{inp} \le T_{out}$: $\varepsilon_{inp} = F_{Pinp} / k_{arm\_inp}$ (refer to Figure 25) It is nonzero only if check box “Elasticity of rocker arms” is active.
lift of output side arm (valve) along follower motion	Output *x*-coordinate (X_{out}) of output side arm end (refer to Figure 19).
lift of output side arm (valve) due to elastic deformation	Deformation (ε_{out}) is calculated according to the formula: for $T_{inp} > T_{out}$: $\varepsilon_{out} = F_{Pout} / k_{arm\_out}$ for $T_{inp} \le T_{out}$: $\varepsilon_{out} = F_{Pinp} \dfrac{L_{CS}}{L_{VS}} / k_{arm\_out}$ (refer to Figure 25) It is nonzero only if check box “Elasticity of rocker arms” is active.
velocity of output side arm (valve) along follower motion	Velocity in output *x*-direction of output side arm end.

rotation angle of rocker arm	Angle of rocker arm motion around its rotation axis. It is used for the calculation of contact point and output *x*-coordinate (α_{RA}), considering adequate corrections due to elastic deformations of arms and supports.
angular velocity of rocker arm around rotation axis	Angular velocity of rocker arm motion around rotation axis ($\dot{\alpha}_{RA}$).
angular acceleration of rocker around rotation axis	Angular acceleration of rocker arm motion around rotation axis: $$\ddot{\alpha}_{RA} = \frac{T_{inp} - T_{out} - T_{frict}}{I_{RA}}$$
support lift along input side cam-roller center axis	At each step the support coordinates in local *x* and *y*-direction are calculated according to the formula: $$\ddot{x}_{su} = \frac{F_{Xsu} - F_{Xbr\_c} - F_{Xbr\_k}}{m}$$ $$\ddot{y}_{su} = \frac{F_{Ysu} - F_{Ybr\_c} - F_{Ybr\_k}}{m}$$ (refer to Figure 16) where: $$F_{Xsu} = F_{norm} \cos(\beta_{press}) + F_{Cout} \cos(\pi - \beta_I)$$ $$F_{Ysu} = F_{norm} \sin(\beta_{press}) - F_{Cout} \sin(\pi - \beta_I)$$ (refer to Figure 16 and Figure 30) The above support coordinates are used for position correction of the contact point between roller and cam profile. Support lift along cam-roller center axis is nonzero only if check box “Elastic rocker support” is active.
support lift along output side arm follower motion	As stated above, at each step the support coordinates in local *x* and *y*-direction are calculated. These coordinates are transferred into the coordinate system of follower motion and further used for the correction of the output *x*-coordinate. Support lift along follower motion is nonzero only if check box “Elastic rocker support” is active.
amplitude of rocker arm support motion	Amplitude of support motion is calculated from the support coordinates in local *x* and *y*-direction: $$A_{su} = \sqrt{x_{su}^2 + y_{su}^2}$$
friction force in rocker arm support bearing	Friction force in the support bearing is calculated from to the formula: $$F_{frict} = F_{frict\_0} + \mu_{br} F_{su} + \lambda_{br} \frac{\omega_{RA}}{R_{br}}$$ It is nonzero only if check box “Friction parameters” is active.

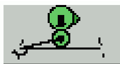
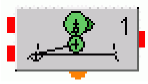
friction torque in rocker arm support bearing	Friction torque in support bearing is calculated from to the formula: $$T_{frict} = sign(\omega_{RA}) F_{frict} R_{br}$$ It is nonzero only if check box “Friction parameters” is active.
reaction force on input arm along cam-roller center axis	Reaction force on input arm (F_{norm}) along initial cam-roller center axis is result of the contact force between roller and cam profile (refer to Figure 24).
reaction force on output arm along follower motion	Reaction force on output arm along follower motion is result of preload, stiffness and damping forces of output mechanical connection.
resultant torque around rocker arm rotation axis	Resultant torque around rotation axis is calculated according to the formula: $$T_{res} = T_{inp} - T_{out} - T_{frict}$$ where: $$T_{inp} = F_{Pinp} L_{CS}$$ $$T_{out} = F_{Pout} L_{VS}$$ (refer to Figure 19 and Figure 25)
maximum torque around rocker arm rotation axis	Maximum torque value from calculation start till actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).
lift of cam center along cam-roller center axis	Input *x*-coordinate of cam center. Nonzero only if **Rocker Arm (Cam)** is attached via input mechanical connections in *x*-direction to e.g. **Mass** or **Shaft** elements (i.e. **Rocker Arm (Cam)** is not a boundary element).
lift of cam center normal to cam-roller center axis	Input *y*-coordinate of cam center. Nonzero only if **Rocker Arm (Cam)** is attached via input mechanical connections in *y*-direction to e.g. **Mass** or **Shaft** elements (i.e. **Rocker Arm (Cam)** is not a boundary element).
amplitude of cam center motion	Amplitude of cam center motion is calculated from coordinates of cam center in input *x* and *y*-direction $$A_{CC} = \sqrt{x_{inp}^2 + y_{inp}^2}$$
rotation angle of cam(shaft) body	Input *w*-coordinate (rotation angle) of cam(shaft). If **Rocker Arm (Cam)** is a boundary element (i.e. an input mechanical connection in *w*-direction does not exist), rotation angle of cam profile is the product of Reference speed $\dot{\varphi}_{ref}$, transmission ratio i_{tr} and time interval from the beginning of calculation t: $$w = \dot{\varphi}_{cam}\, t = i_{tr} \dot{\varphi}_{ref}\, t\ .$$ If an input mechanical connection in *w*-direction exists, then rotation angle is calculated from the dynamics of the system.

angular velocity of cam body rotation	Angular velocity in *w*-direction of cam(shaft) rotation. If **Rocker Arm (Cam)** is a boundary element (i.e. it does not have any input mechanical connection in *w*-direction), its angular velocity is equal to Reference speed multiplied by transmission ratio. Otherwise angular velocity of cam(shaft) body is calculated at each step from the equations of motion of the system.
pressure angle of cam profile	Pressure angle of cam profile (refer to β_{press} in Figure 30). It is an angle between the normal force on cam profile and straight line connecting initial positions of cam center and roller center.
reaction force normal to cam profile	Force normal to cam profile F_{norm} (refer to Figure 30). It is calculated according to the formula: $$F_{norm} = F_k + F_c$$ where: $$F_k = \sqrt[3]{\frac{512\varepsilon_{cont}^4 E_{cam}^3 b_R R_R}{81\pi\left(1-\nu_{cam}^2\right)^3}}$$ $$k_{cont} = \frac{F_k}{\varepsilon_{cont}}$$ $$c_{cont} = r_c 2\sqrt{m_{RA} k_{cont}}$$ $$F_c = v_{cont} c_{cont}$$
torque of reaction force around cam rotation axis	Torque of reaction force around cam rotation axis is calculated according to the formula: $$T_{react} = F_{norm} X \sin\left(\pi - \alpha_{cp} - \beta_{cp}\right)$$  **Figure 31: Distance between cam and roller centers**

radius of curvate of cam profile	Radius of curvature of cam profile corresponds to radius R_{prof} (refer to Figure 2 in Section 4.1.1).
Hertz stress at cam-roller contact	Hertz stress at cam-roller contact is calculated from the formula $$p_{Hertz} = 0.418\sqrt{\frac{E_{cam}F_{norm}}{b_{roll}}\left(\frac{1}{R_{prof}}+\frac{1}{r_{roll}}\right)}.$$

maximum Hertz stress till actual point	Maximum Hertz stress value from calculation start till actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).

5.3. Finger Follower

Element Name:	**Finger Follower**	
Element Icon:		
Definition:	This element serves to define the properties of Finger Follower.	
Connecting pins:	standard pins: 3 (mechanical) special pins: 0 electrical pins: 1 Two input mechanical connections in x and y-direction can be specified. Only one output mechanical connection in x-direction can be specified.	

Note: Currently the Variable-step solver does not support this element.

The schematic of **Finger Follower** is shown in Figure 32.

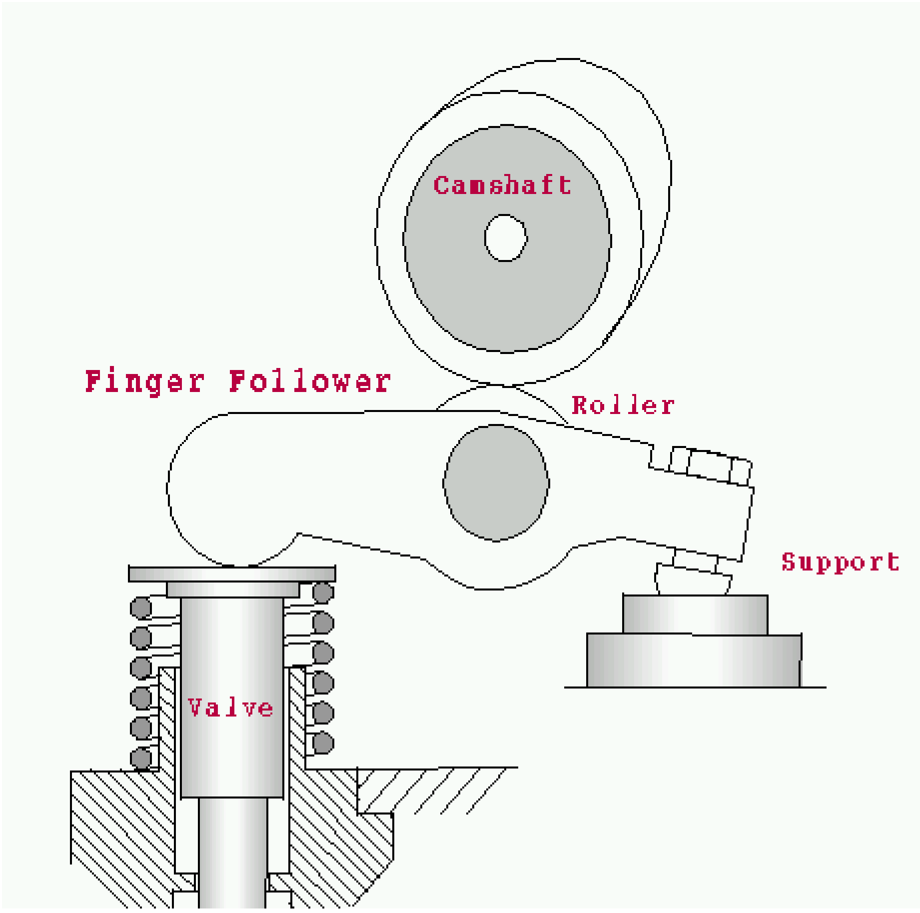


Figure 32: Finger Follower Schematic

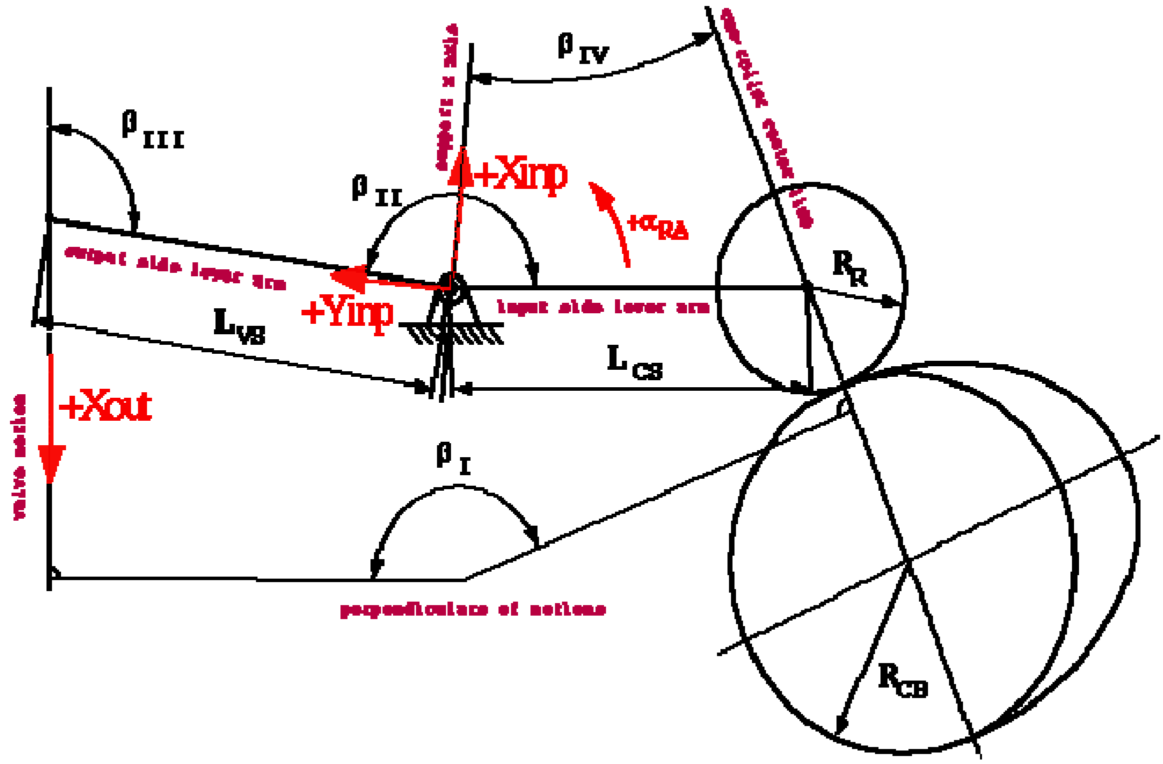


Figure 33: Finger Follower Geometry

5.3.1. Input Parameters

Description of input data for the **Finger Follower**:

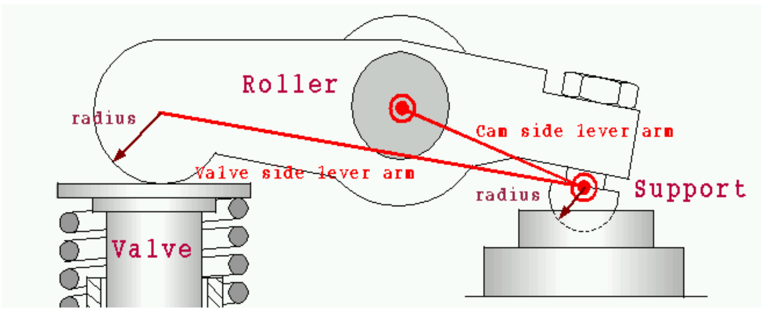
<table>
<tr><td>Element name:</td><td>The name of the element is specified as default.</td></tr>
<tr><td>Cam Profile</td><td>Press this bar to specify the cam profile in polar coordinates and characteristics of contact between roller and cam body (refer to Section 5.2.2).</td></tr>
<tr><td>Elastic Camshaft</td><td>Press this bar to specify the characteristics of cam shaft support and torsion (refer to Section 5.3.2).</td></tr>
<tr><td colspan="2">Inertia data</td></tr>
<tr><td>Finger follower mass</td><td>Specify mass of finger follower with roller (m_{FF}).</td></tr>
<tr><td>Moment of inertia about rotation axis</td><td>Specify moment of inertia of rocker arm with roller about rotation axis (I_{FF}).</td></tr>
<tr><td colspan="2">Geometric data</td></tr>
<tr><td>Length of cam side lever arm (support-roller)</td><td>Specify length of cam side lever arm (L_{CS}) (Figure 33). Cam side lever arm is line between contact arc center of support and center of roller rotation.


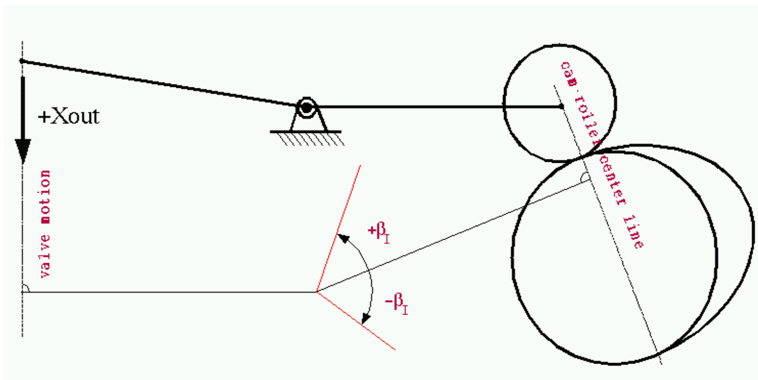
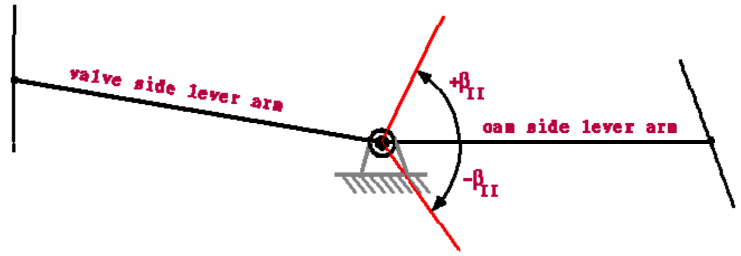
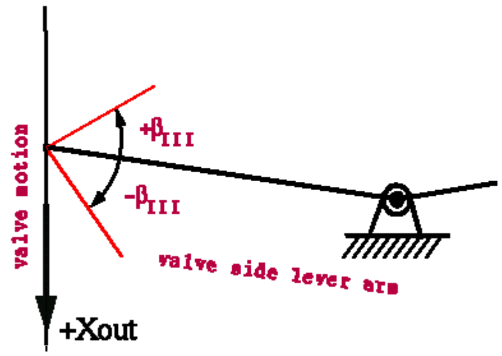
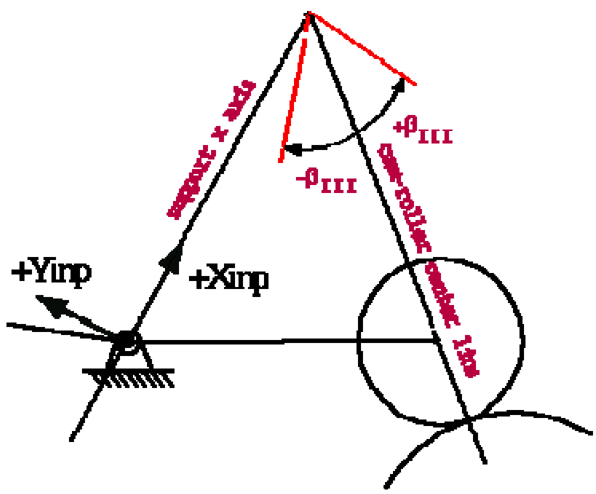
Figure 34: Cam and Valve side lever arms</td></tr>
<tr><td>Length of valve side lever arm (support-finger)</td><td>Specify length of valve side lever arm (L_{VS}) (refer to Figure 33). Valve side lever arm is line between contact arc center of support and contact arc center on output side (refer to Figure 34).</td></tr>
<tr><td>Angle between perpendiculars of motion</td><td>Specify angle between perpendiculars on cam-roller center line and positive valve motion (β_I).


Figure 35: Angle between perpendiculars of motions</td></tr>
</table>

Angle between cam and valve lever arms	Specify angle between cam and valve lever arms (β_{II}).  **Figure 36: Angle between cam and valve lever arms**
Angle between valve side arm/valve motion	Specify angle between valve side arm and negative valve motion (β_{III}).  **Figure 37: Angle between valve side arm and valve motion**
Angle between cam-roller centerline/ support x-axis	Specify angle between roller-cam centerline and negative input motion (support x-axis) (β_{III}).  **Figure 38: Angle b. cam-roller centerline and support x-axis**
Elasticity of lever arms	If check box is inactive (or stiffness is zero), elastic deformation of respective lever arm will not be calculated.

Stiffness normal to input side arm (cam)	Specify stiffness normal to cam side arm (k_{arm_inp}). This stiffness is applied for the calculation of elastic deformation of cam side arm. Calculated value is used in correction of roller position in contact with cam profile (refer to Figure 39 and Figure 40).
Stiffness normal to output side arm (valve)	Specify stiffness normal to valve side arm (k_{arm_out}). This stiffness is applied for the calculation of elastic deformation of valve side arm. Calculated value is used in correction of element output x-coordinate (X_{out}) (refer to Figure 33).

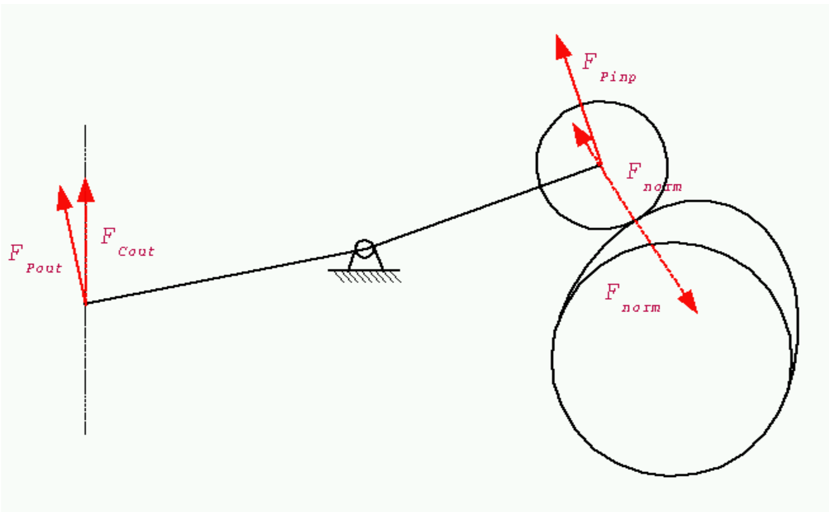


Figure 39: Forces on Finger Follower

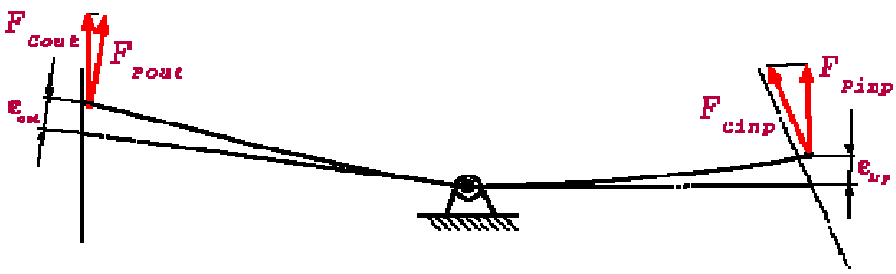


Figure 40: Elasticity of lever arms

Finger Support Friction	Press this bar to specify the friction parameter inside bearing. (refer to Section 5.2.3).

5.3.2. Elastic Camshaft

This **Input Data Dialog Box** of **Elastic Camshaft** serves to define the characteristics of support of cam body or connection to camshaft.

Description of input data:

Elastic support	If check box is inactive (or camshaft mass is zero), support motion in local x and y-direction will not be calculated.
Cam(shaft) mass	Specify mass of camshaft (m_{CS}) (refer to Figure 41).
Support bearing stiffness	Specify support bearing stiffness (k_{br}) (refer to Figure 41).
Support bearing	Specify support bearing damping (c_{br}) (refer to Figure 41).

damping	
Camshaft torsion	If check box is inactive (or camshaft moment of inertia is zero), camshaft torsion will not be calculated.
Moment of inertia	Specify moment of inertia of cam shaft (I_{CS}) (refer to Figure 41).
Torsional stiffness	Specify torsional stiffness (k_{wbr}) (refer to Figure 41).
Torsional damping	Specify torsional damping (c_{wbr}) (refer toFigure 41).

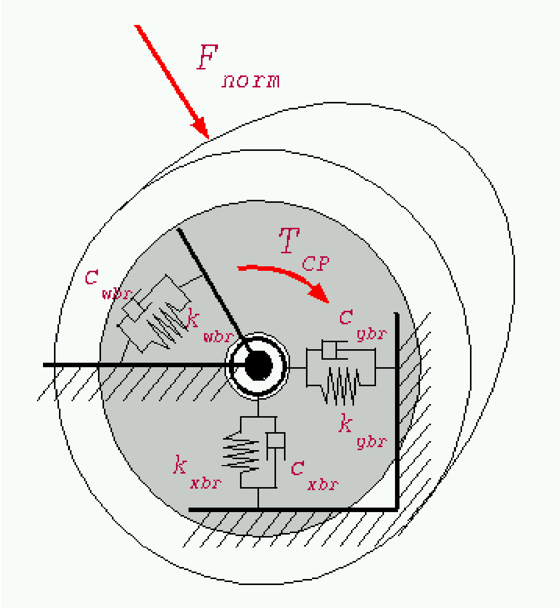


Figure 41: Elastic camshaft support

5.3.3. Finger Support Friction

This **Input Data Dialog Box** of **Finger Support Friction** serves to define the characteristics of friction inside finger support.

Description of input data:

Friction parameters	If check box is inactive (or bearing diameter is zero), friction data input is disabled and friction torque will not be calculated.
Bearing (shaft) diameter	Specify bearing diameter (R_{br}) (refer to Figure 42).
Bearing friction force	Specify bearing friction force (F_{frict_0}) (refer to Figure 42).
Coulomb friction coefficient	Specify friction coefficient (μ_{br}) (refer to Figure 42).
Viscous friction coefficient	Specify viscous friction coefficient (λ_{br}).

Friction torque is calculated according to the formula:

$$T_{frict} = sign(\omega_{RA})\left(F_{frict\_0} + \mu_{br}F_{su} + \lambda_{br}\frac{\omega_{FF}}{R_{br}}\right)R_{br}$$

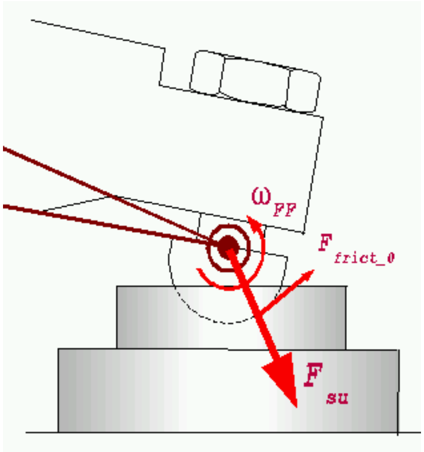


Figure 42: Finger Support Friction

5.3.4. Initial Conditions

Path: **Element** | **Initial Conditions**

Description of initial conditions:

Actuator Side (input)	
Initial translation	
x-coordinate	Specify the initial position of finger follower support in input x-direction.
velocity in x-direction	Specify the initial velocity of finger follower support in input *x*-direction.
y-coordinate	Specify the initial position of finger follower support in input y-direction.
velocity in y-direction	Specify the initial velocity of finger follower support in input y-direction.
Valve Side (output)	
Initial translation	
x-coordinate	Specify the initial position of valve side arm end in output x-direction
velocity in x-direction	Specify the initial velocity of valve side arm end in output x-direction.

5.3.5. Modifiable Parameter

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is checked, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

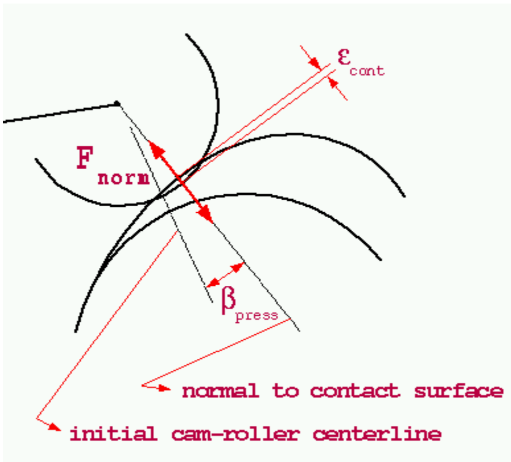
finger follower mass	SI Unit:	kg
moment of inertia about support axis	SI Unit:	kg/m^2
stiffness normal to cam side lever arm	SI Unit:	N/m
stiffness normal to valve side lever arm	SI Unit:	N/m
stiffness of cam(shaft) support bearing	SI Unit:	N/m
damping of cam(shaft) support bearing	SI Unit:	Ns/m
torsional stiffness of camshaft	SI Unit:	Nm/rad
torsional damping of camshaft	SI Unit:	Nms/rad
friction force in cam(shaft) bearing	SI Unit:	N
Coulomb friction coefficient in bearing	SI Unit:	-
viscous friction coefficient in bearing	SI Unit:	Ns/m

5.3.6. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter must be checked.

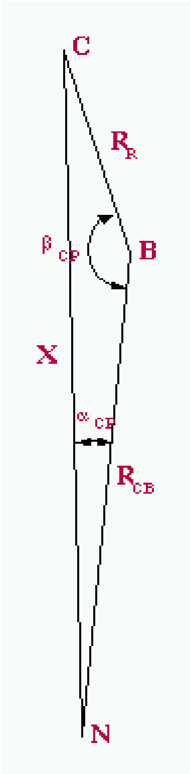
Description of output parameters:

cam profile (contact point) lift along cam-roller center axis	This is a projection of the actual polar coordinate of cam profile (distance between contact point and center of cam body rotation) to cam-roller center axis.
elastic deformation at contact point (normal to cam profile)	Elastic deformation at contact point is calculated from the position of finger support, finger angle, angle of cam body and position of cam body center. This deformation is used for the calculation of the contact force.  **Figure 43: Elastic Deformation at Contact Point**

elastic deformation of cam side arm (normal to cam profile)	Deformation (ε_{inp}) is calculated according to the formulas: for $T_{inp} > T_{out}$: $\varepsilon_{inp} = F_{Pout} \frac{L_{VS}}{L_{CS}} / k_{arm\_inp}$ for $T_{inp} \le T_{out}$: $\varepsilon_{inp} = F_{Pinp} / k_{arm\_inp}$ (refer to Figure 33 and Figure 40) It is nonzero only if check box “Elasticity of lever arms” is active.
x-coordinate of cam support bearing center (input side)	Input x-coordinate of finger follower support (refer to Figure 33). Nonzero only if **Finger Follower** has an input mechanical connection in x-direction (i.e. **Finger Follower** is not a boundary).
y-coordinate of cam support bearing center (input side)	Input y-coordinate of finger follower support (refer to Figure 33). Nonzero only if **Finger Follower** has an input mechanical connection in y-direction (i.e. **Finger Follower** is not a boundary).
amplitude of cam support bearing motion (input side)	Amplitude of support motion is calculated from coordinates of support center in input x and y-direction: $A_{SC} = \sqrt{x_{inp}^2 + y_{inp}^2}$
lift of output side arm (valve) along valve motion	Output x-coordinate (X_{out}) of valve side arm end (refer to Figure 33).
lift of output side arm (valve) due to elastic deformation	Deformation (ε_{out}) is calculated according to the formulas: for $T_{inp} > T_{out}$: $\varepsilon_{out} = F_{Pout} / k_{arm\_out}$ for $T_{inp} \le T_{out}$: $\varepsilon_{out} = F_{Pinp} \frac{L_{CS}}{L_{VS}} / k_{arm\_out}$ (refer to Figure 33 and Figure 40) It is nonzero only if check box “Elasticity of lever arms” is active.
velocity of output side arm (valve) along valve motion	Velocity in output x-direction of valve side arm end.
rotation angle of finger follower around support rotation axis	Angle of finger motion around support rotation axis. It is used for the calculation of contact point and output *x*-coordinate (α_{FF}), considering adequate corrections due to elastic deformations of arms and supports.
angular velocity of finger follower around support axis	Angular velocity of finger follower around support rotation axis ($\dot{\alpha}_{FF}$).
angular acceleration of finger follower around support axis	Angular acceleration of finger follower around support rotation axis: $\ddot{\alpha}_{FF} = \frac{T_{inp} - T_{out} - T_{frict}}{I_{FF}}$
reaction force on cam side arm along input support x-axis	Reaction force on cam side arm (F_{norm}) along input support x-coordinate is result of contact force between roller and cam profile (refer to Figure 40).

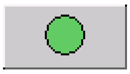
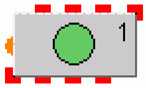
reaction force on valve side arm along output valve motion	Reaction force on valve side arm along valve motion is result of preload, stiffness and damping forces of output mechanical connection.
resultant torque around finger follower rotation axis	Resultant finger torque around rotation axis is calculated according to the formula: $$T_{res} = T_{inp} - T_{out} - T_{frict}$$ where: $$T_{inp} = F_{Pinp} L_{CS}$$ $$T_{out} = F_{Pout} L_{VS}$$ (refer to Figure 33 and Figure 40)
maximum torque around finger follower rotation axis	Maximum torque value from calculation start till actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).
friction force in finger support	Friction force in cam(shaft) bearing is calculated according to the formula (refer to Figure 42): $$F_{frict} = F_{frict\_0} + \mu_{br} F_{su} + \lambda_{br} \frac{\omega_{FF}}{R_{br}}$$ It is nonzero only if check box “Friction parameters” is active.
friction torque in finger support	Friction torque in cam(shaft) support bearing is calculated according to the formula: $$T_{frict} = sign(\omega_{RA}) F_{frict} R_{br}$$ It is nonzero only if check box “Friction parameters” is active.
lift of cam center along cam-roller center axis	At each step the local *x*-coordinate of cam(shaft) support is calculated from the formula (refer to Figure 41): $$\ddot{x}_{CSsu} = \frac{F_{normX} - F_{Xbr\_c} - F_{Xbr\_k}}{m_{CS}}$$ Lift of cam center along cam-roller center axis is nonzero only if check box “Elastic Camshaft support” is active.
lift of cam center normal to cam-roller center axis	At each step the local *y*-coordinate of cam(shaft) support is calculated from the formula (refer to Figure 41): $$\ddot{y}_{CSsu} = \frac{F_{nromY} - F_{Ybr\_c} - F_{Ybr\_k}}{m_{CS}}$$ Cam(shaft) support coordinates are used for position correction of the contact point between roller and cam profile. Lift of cam center normal to cam-roller center axis is nonzero only if check box “Elastic Camshaft support” is active.

amplitude of cam center motion	Amplitude of cam center motion is calculated from the coordinates of cam center in local *x* and *y*-direction: $$A_{CC} = \sqrt{x_{CSsu}^2 + y_{CSsu}^2}$$
rotation angle of cam body around cam(shaft) axis	*w*-coordinate (rotation angle) of cam body
angular velocity of cam body rotation	Angular velocity in *w*-direction of cam body rotation.
pressure angle of cam profile	Pressure angle of cam profile (refer to β_{press} in Figure 43). It is an angle between the normal force on cam profile and straight line connecting initial positions of cam center and roller center.
reaction force normal to cam profile	Force normal to cam profile F_{norm} (refer to Figure 43). It is calculated according to the formula: $$F_{norm} = F_k + F_c$$ where: $$F_k = \sqrt[3]{\frac{512\varepsilon_{cont}^4 E_{cam}^3 b_R R_R}{81\pi\left(1-\nu_{cam}^2\right)^3}}$$ $$k_{cont} = \frac{F_k}{\varepsilon_{cont}}$$ $$c_{cont} = r_c 2\sqrt{m_{RA} k_{cont}}$$ $$F_c = v_{cont} c_{cont}$$

torque of reaction force around cam rotation axis	Torque of reaction force around cam rotation axis is calculated according to the formula: $$T_{react} = F_{norm} X \sin(\pi - \alpha_{cp} - \beta_{cp})$$  **Figure 44: Distance between cam and roller centers**
radius of curvate of cam profile	Radius of curvature of cam profile corresponds to radius R_{prof} (refer to Figure 2 in Section 4.1.1).
Hertz stress at cam-roller contact	Hertz stress at cam-roller contact is calculated from the formula $$p_{Hertz} = 0.418 \sqrt{\frac{E_{cam} F_{norm}}{b_{roll}} \left(\frac{1}{R_{prof}} + \frac{1}{r_{roll}} \right)}.$$
maximum Hertz stress till actual point	Maximum Hertz stress value from calculation start until actual time/angle step. It is useful for e.g. Search/Adjust Parameter option (1D Optimisation).

6. SOLID

6.1. Mass

Element Name:	**Lumped Mass**	
Element Icon:		
Definition:	This element serves to define the lumped Mass (with two DOFs).	
Connecting pins:	standard pins: 8 (all mechanical) special pins: 0 wire pins: 1	

Note: **Mass** element may have mechanical connections only in x and y-directions. Multiple connections to the same element are not allowed. If specified, all connections except the first will be ignored.
Mass element has two DOFs: translational motion in x and y-directions. No rotation can be considered.

Note: For **Mass** element, input parameters and initial conditions have to be defined in local x-y coordinate system. Output parameters as coordinates, velocities and accelerations are also calculated in the local coordinate system.

Note: Local x and y-axis of **Mass** may only be **translated** in the global x-y coordinate system. There is no angle between **Mass** local x-direction and global x-direction.

Dynamic model of lumped **Mass** element is shown in Figure 45.

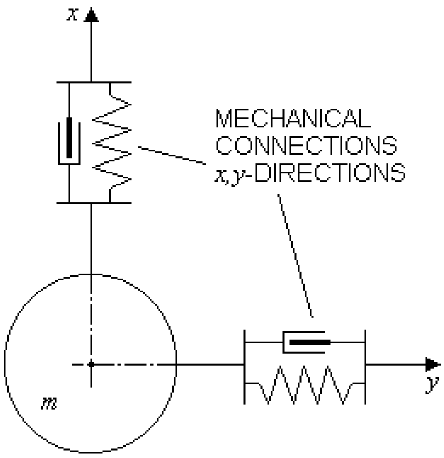


Figure 45: Lumped Mass with Connections

6.1.1. Input Parameters

Description of input data for Lumped **Mass**:

Element name:	The name of the element is specified as default.
Moving mass	Specify moving mass. It is equal to actual mass + 33% of mass of the attached springs (connections).
Coulomb friction force in x-direction	Specify the Coulomb friction force in x-direction.
Coulomb friction force in y-direction	Specify the Coulomb friction force in y-direction.

6.1.2. Initial Conditions

Path: **Element** | **Initial Conditions**

Description of initial conditions:

x-coordinate	Specify the initial position in x-direction.
velocity in x-direction	Specify the initial velocity in x-direction.
y-coordinate	Specify the initial position in y-direction.
velocity in y-direction	Specify the initial velocity in y-direction.

All initial values that are not specified in the dialog box are set to 0.

6.1.3. Modifiable Parameters

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name must be checked. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameter:

Moving mass	SI units:	kg
Coulomb friction force in x-direction	SI units:	N
Coulomb friction force in y-direction	SI units:	N

☞ **Note:** Modification implies explicit parameter variation within the calculation. This should be used with care.

6.1.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button on the left of output parameter name has to be checked.

List of output parameters:

x-coordinate of gravity center
velocity of gravity center in x-direction
acceleration in x-direction
y-coordinate of gravity center
velocity of gravity center in y-direction
acceleration in y-direction
resultant force in x-direction
resultant force in y-direction

At high connecting stiffness and small values of the moving mass high natural frequencies will occur in the system, which may cause numerical problems for the integration. To avoid numerical instability, a very short time step has to be specified in this case.

The governing equations of motion are explained in Section 6.1.5. A short theoretical background can be found in Section 6.2.9.

6.1.5. Equations of Motion

The motion of Lumped Mass is governed by the following equations (coordinate system of element and local coordinate systems of connected elements and connections are only translated one to each other):

$$m\ddot{x}+\sum_{i=1}^{n_x}c_{xi}(\dot{x}-\dot{x}_i)-\sum_{j=1}^{l_x}c_{xj}(\dot{x}-\dot{x}_j)+\sum_{i=1}^{n_x}k_{xi}(x-x_i)-\sum_{j=1}^{l_x}k_{xj}(x-x_j)=\sum_{i=1}^{n_x}F_{0xi}-\sum_{j=1}^{l_x}F_{0xj}-F_{xfrict},$$

$$m\ddot{y}+\sum_{i=1}^{n_y}c_{yi}(\dot{y}-\dot{y}_i)-\sum_{j=1}^{l_y}c_{yj}(\dot{y}-\dot{y}_j)+\sum_{i=1}^{n_y}k_{yi}(y-y_i)-\sum_{j=1}^{l_y}k_{yj}(y-y_j)=\sum_{i=1}^{n_y}F_{0yi}-\sum_{j=1}^{l_y}F_{0yj}-F_{yfrict}, \tag{48}$$

where

m	mass
x and y	coordinates of Mass element
x_i and y_i	coordinates of *i-th* connected elements on input end
x_j and y_j	coordinates of the *j-th* connected elements on output end
c_{xi} and c_{xj}	damping coefficients (functions) of the *i-th* input and *j-th* output mechanical connections in *x-direction*
k_{xi} and k_{xj}	stiffness coefficients (functions) of the *i-th* input and *j-th* output mechanical connections in *x-direction*
c_{yi} and c_{yj}	the damping coefficients (functions) of the *i-th* input and *j-th* output mechanical connections in *y-direction*
k_{yi} and k_{yj}	the stiffness coefficients (functions) of the *i-th* input and *j-th* output mechanical connections in *y-direction*
F_{0xi} and F_{0xj}	preload forces of the *i-th* input and *j-th* output mechanical connections in *x-direction*

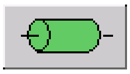
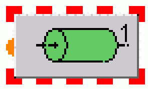
F_{0yi} and F_{0yj}	preload forces of the *i-th* input and *j-th* output mechanical connections in *y-direction*
n_x and l_x	number of mechanical connections on input and output ends in *x-direction*
n_y and l_y	number of mechanical connections on input and output ends in *y-direction*
F_{xfrict} and F_{yfrict}	Coulomb friction forces in *x* and *y-directions,* respectively

Damping and stiffness coefficients (functions) and preload forces in Equation 48 are specified in the input dialogs of respective mechanical connections (refer to Chapter 21).

For constant damping and stiffness coefficients of mechanical connections, Equation 48 is linear. However, if at least one mechanical connection is variable, Equation 48 would be non-linear. Variable connection implies that stiffness is a function of relative displacement and/or damping is a function of relative displacement or velocity.

☞ **Note:** If the coordinate system of **Mass** element and local coordinate systems of connected elements and connections are not translated but there is an angle between their local x-directions (some of them are rotated with respect to the global coordinate system), see Section 6.2.9.

6.2. Shaft

Element Name:	**Shaft**	
Element Icon:		
Definition:	This element serves to define a rigid or elastic Shaft.	
Connecting pins:	standard pins: 10 (all mechanical) special pins: 0 wire pins: 1	

☞ **Note:** **Shaft** element may have mechanical connections in x, y and w-directions, respectively. Connections in x and y-directions are linear (translational springs and/or dampers), while connections in w direction define rotational motion (torsional springs and/or dampers) as shown in Figure 46. Multiple connections to the same element are not allowed. If specified, all connections except the first will be ignored.
Rigid **Shaft** element has three degrees-of-freedom (DOFs): translation in x and y-direction and rotation about w axis. Elastic **Shaft** has four DOFs: translation in x and y-direction and rotation around w axis for input and output end, separately.

Note: For the **Shaft** element, input parameters and initial conditions have to be defined in local x-y coordinate system. Output parameters such as coordinates, velocities and accelerations are also calculated in the local coordinate system.

Note: Local x and y-axis of **Shaft** may only be **translated** in the global x-y coordinate system. There is no angle between **Shaft** local x-direction and global x-direction.

For modelling of mechanical drives of injection pumps and valve trains, the **Shaft** has to be connected to the input side of **Cam Profile** element (refer to Chapter 4). Normally, the mechanical connections in all three directions (x, y and w) have to be defined there. For modelling the drive of the distributor-type injection pump, **Shaft** must be attached to the input side of **Cam Plate** element as shown in Chapter 4. Only connection (torsional spring/damper) in w-direction is necessary. Dynamic model of **Shaft** with mechanical connections is shown in Figure 46.

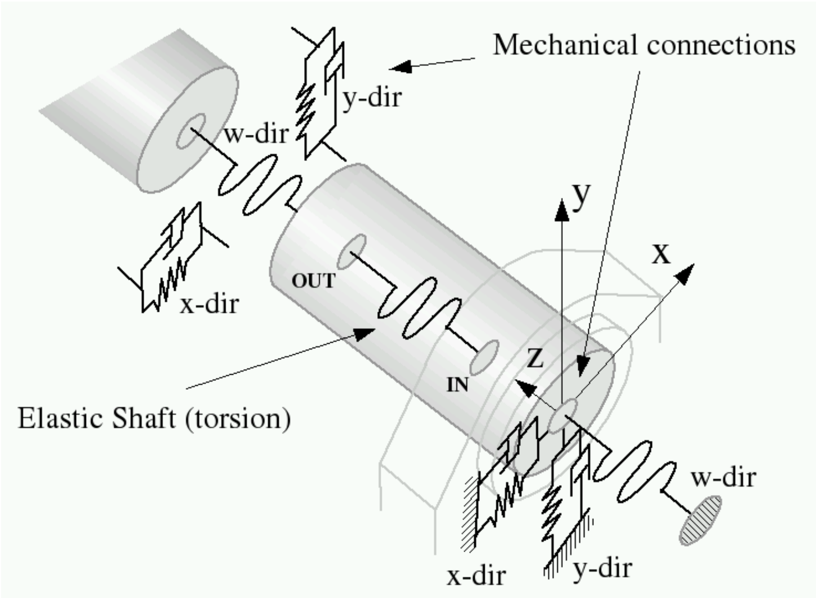


Figure 46: Shaft Element with Connections

6.2.1. Input Parameters

Description of input data for the **Shaft**:

Element name:	The name of the element is specified as default.
Moving mass	Specify moving mass. It is equal to the actual shaft mass + 33% of mass of the attached spring. Mass may be set to 0 (no inertia) if only torsional vibration has to be considered (no connections in x and y- directions defined).
Coulomb friction force at translation	Specify the Coulomb friction force in x and y-direction. The same force is applied for both directions.
Rigid Shaft	**Rigid Shaft** is a rigid body with 3 DOFs, i.e. without torsional elasticity between the input and output end of **Shaft** element.
Moment of inertia about axis w	Specify moment of inertia around rotational axis *Ip.* Contrary to the moving mass, moment of inertia must be specified (nonzero).
Coulomb friction torque at rotation	Specify Coulomb friction torque around rotation axis w.
Elastic Shaft (torsion)	**Elastic Shaft** implies an elastic body with the torsional elasticity between input and output side of **Shaft** element. Stiffness and damping characteristics of the elastic shaft are calculated according to the geometric and material data.

Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the elastic shaft (refer to section 6.2.2).
Static UNBALANCE (OPTIONAL)	

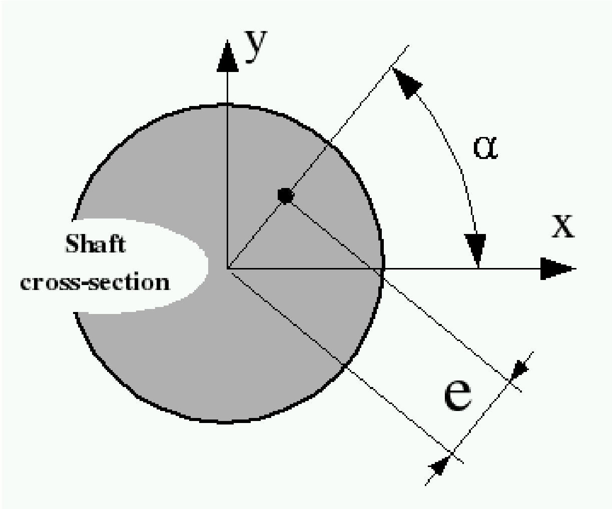


Figure 47: Eccentricity of Gravity Centre

Eccentricity of gravity center	Specify the distance from shaft center of gravity to rotation axis w (parameter e in Figure 47).
Angle to x axis at calculation start	Specify initial angle between x axis and straight line drawn from the center of gravity to the geometric center (α in Figure 46).
According to eccentricity of gravity center relative to rotation axis w (defined with distance e and angle α), BOOST Hydsim will calculate additional centrifugal forces in xy-plane. These forces will be considered as additional forces in the equations of motion of **Shaft** element (refer to Section 6.2.9).	

6.2.2. Elastic Shaft Properties

Elastic shaft option allows to calculate the torsional deformation of the shaft as a result of the torques acting on its input and output end. Stiffness k_{ts} and damping c_{ts} of the virtual spring and damper represent the torsional elasticity between the input and output end as shown in Figure 48.

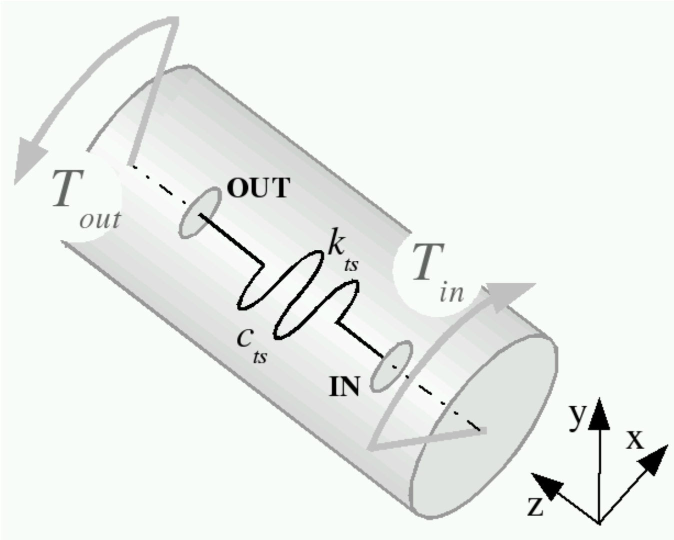
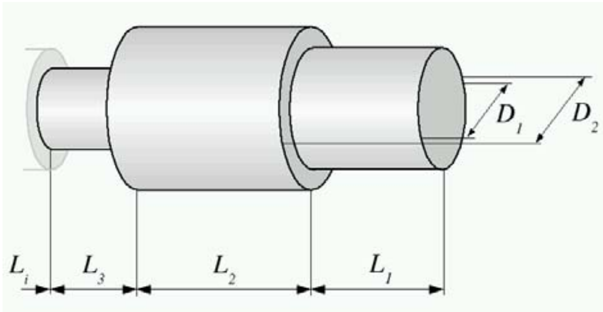
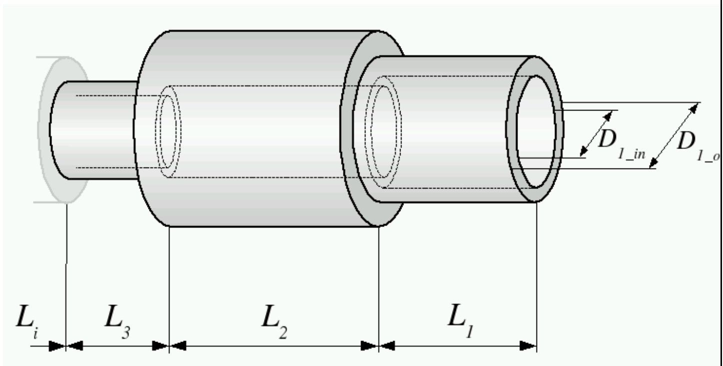


Figure 48: Mechanical Model of Elastic Shaft

For the internal calculation of the torsional stiffness k_{ts} and damping c_{ts}, full geometry and material data of the shaft piece have to be specified in the dialog **Elastic Shaft**.

Description of input data:

Material Data	
Solid Properties …	Press this bar to define the local solid properties (refer to Section 8.2.2). Global solid properties are active by default.
Equivalent torsional damping ratio	Specify equivalent viscous damping ratio for torsion (r_C). Default value is set to critical damping.

Geometric Data	
Cross-section type:	
Solid	Geometry and input data of the solid shaft is shown in Figure 49.
Cross-section table	Specify the diameters D_i and lengths L_i of all shaft sections.  **Figure 49: Solid Shaft Geometry** Torsional stiffness of the shaft k_{ts} is calculated from the torsional stiffness of each shaft section k_i: $\frac{1}{k_{ts}} = \sum_i \frac{1}{k_i}$, where $k_i = \frac{GJ_{Pi}}{L_i}$. Polar moment of area and shear modulus are given by: $J_{Pi} = \frac{\pi D_i^4}{32}$, $G = \frac{E}{2(1+\mu)}$. Torsional damping of the shaft *cts* is calculated from the critical damping c_{t_crit}: $c_{t\_crit} = 2.\sqrt{I_p k_{ts}}$, $c_{ts} = r_c c_{t\_crit}$, where $I_p = \frac{mD_i^2}{8}$.
Hollow	Geometry and input data of the hollow shaft is shown in Figure 50.
Cross-section table	Specify the inner and outer diameters D_{i_in} and D_{i_out}, respectively, and lengths L_i of all shaft sections.  **Figure 50: Hollow Shaft Geometry**

	Torsional stiffness of the shaft k_{ts} is calculated from the torsional stiffness of each shaft section k_i: $$\frac{1}{k_{ts}} = \sum_i \frac{1}{k_i}, \quad \text{where} \quad k_i = \frac{GJ_{Pi}}{L_i}.$$ Polar moment of area and shear modulus are given by: $$J_{Pi} = \frac{\pi\left(D_{i\_out}^4 - D_{i\_in}^4\right)}{32}, \quad G = \frac{E}{2(1+\mu)}.$$ Torsional damping of the shaft c_{ts} is calculated from the critical damping c_{t_crit}: $$c_{t\_crit} = 2.\sqrt{I_p k_{ts}}$$ $$c_{ts} = r_c c_{t\_crit}, \text{ where } I_p = \frac{m\left(D_{i\_out}^2 - D_{i\_in}^2\right)}{8}.$$
Additional moment of inertia	
Inertia moment at input end	Specify additional (rigid) moment of inertia at input end I_{add_inp} .
Inertia moment at output end	Specify additional (rigid) moment of inertia at output end I_{add_out}
Coulomb friction torque	
Friction torque at input end	Specify additional Coulomb friction torque around rotation axis w at input end T_{C_inp}.
Friction torque at output end	Specify additional Coulomb friction torque around rotation axis w at output end T_{C_out}.

6.2.3. Local Solid Properties

This dialog serves to define the local solid properties.
Local solid properties can be **constant** or **variable** (temperature-dependent).

Description of input data:

GLOBAL SOLID PROPERTIES	Solid properties are taken from the table of **Global Solid Properties** dialog box.
SOLID PROPERTIES FROM PROPERTY DATABASE	Click to activate selection of solid properties from Property Database (refer to Chapter **26.2.4.8** for more information).
Solid name	Specify Solid name from the list.
Temperature	Specify constant solid temperature.

6.2.4. Initial Conditions

Path: **Element**| **Initial Conditions**

All initial values that are not specified in this dialog are set to 0. Description of initial conditions is in Table 5.

Table 5: Initial Values of Rigid Shaft

Initial translation	
x-coordinate	Specify the initial position of gravity center in x-direction.
velocity in x-direction	Specify the initial velocity of gravity center in x-direction.
y-coordinate	Specify the initial position of gravity center in y-direction.
velocity in y-direction	Specify the initial velocity of gravity center in y-direction.
Initial rotation	
Angle	
input w	Specify the initial angle of shaft rotation at input end around axis w.
output w	Specify the initial angle of shaft rotation at output end around axis w. **Note:** If **Rigid Shaft** option is active, then initial angles at input and output end must be the same.
Angular Velocity	
input ω	Specify the initial angular velocity of input end around rotation axis w.
output ω	Specify the initial angular velocity of output end around rotation axis w.
	Note: If **Rigid Shaft** option is active, initial angular velocities at input and output end must be the same.

6.2.5. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be checked. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable Parameters:

Moving mass	SI Units:	kg
Moment of inertia about rotation axis	SI Units:	kgm^2
Coulomb friction force at translation	SI Units:	N
Coulomb friction about rotation axis	SI Units:	Nm
Distance from gravity center to rotation axis	SI Unit:	m

6.2.6. Output Parameters

Path: **Element**| **Store Results…**

To activate output parameter, the option button in front of output parameter has to be checked.

List of output parameters:

x-coordinate of geometry center
velocity of geometry center in x-direction
y-coordinate of geometry center
velocity of geometry center in y-direction
rotation angle (w-coordinate) (input side)
angular velocity in w-direction (input side)
angular acceleration in w-direction (input side)
x-coordinate of gravity center
velocity of gravity center in x-direction
acceleration of gravity center in x-direction
y-coordinate of gravity center
velocity of gravity center in y-direction
acceleration of gravity center in y-direction
resultant force in x-direction
resultant force in y-direction
resultant torque in w-direction
unbalance force in x-direction
unbalance force in y-direction
unbalance torque in w-direction
maximum torque till selected point
rotation angle (w-coordinate) (output side)
angular velocity in w-direction (output side)
angular acceleration in w-direction (output side)

☞ **Note:** If no eccentricity is defined, gravity center coincides with the geometry center and the respective output parameters will be (in pairs) the same.

At high connecting stiffness and small values of the shaft mass, high natural frequencies will occur in the system. These may cause numerical problems for the integration. To avoid numerical instability, a very short time step has to be specified in this case.

At high torsional stiffness and small values of the moment of inertia, high torsional natural frequencies will occur in the system. To avoid numerical instability during integration, a very short time step has to be used.

6.2.7. Equations of Motion

The motion of **Rigid Shaft** is governed by the equations 49 (coordinate system of element and local coordinate systems of connected elements and connections are only translated one to each other).

$$
\begin{aligned}
&m\ddot{x}+\sum_{i=1}^{n_x} c_{xi}(\dot{x}-\dot{x}_i)-\sum_{j=1}^{l_x} c_{xj}(\dot{x}-\dot{x}_j)+\sum_{i=1}^{n_x} k_{xi}(x-x_i)-\sum_{j=1}^{l_x} k_{xj}(x-x_j)=\sum_{i=1}^{n_x} F_{0xi}-\sum_{j=1}^{l_x} F_{0xj}+F_{x\_ext},\\
&m\ddot{y}+\sum_{i=1}^{n_y} c_{yi}(\dot{y}-\dot{y}_i)-\sum_{j=1}^{l_y} c_{yj}(\dot{y}-\dot{y}_j)+\sum_{i=1}^{n_y} k_{yi}(y-y_i)-\sum_{j=1}^{l_y} k_{yj}(y-y_j)=\sum_{i=1}^{n_y} F_{0yi}-\sum_{j=1}^{l_y} F_{0yj}+F_{y\_ext},\\
&I\ddot{w}+\sum_{i=1}^{n_w} c_{wi}(\dot{w}-\dot{w}_i)-\sum_{j=1}^{l_w} c_{wj}(\dot{w}-\dot{w}_j)+\sum_{i=1}^{n_w} k_{wi}(w-w_i)-\sum_{j=1}^{l_w} k_{wj}(w-w_j)=\sum_{i=1}^{n_w} T_{0i}-\sum_{j=1}^{l_w} T_{0j}+T_{ext},
\end{aligned}
\tag{49}
$$

For **Elastic Shaft,** the third equation (in w-direction) is replaced with the set of equations:

$$
\begin{aligned}
&\left(I_{p\_inp}+I_{add\_inp}\right)\ddot{w}_{inp}+\sum_{i=1}^{n_w} c_{wi}(\dot{w}_{inp}-\dot{w}_i)+\sum_{i=1}^{n_w} k_{wi}(w_{inp}-w_i)=\sum_{i=1}^{n_w} T_{0i}+0.5T_{ecc}-T_{c\_inp}+T_{s\_stiff}+T_{s\_damp},\\
&\left(I_{p\_out}+I_{add\_out}\right)\ddot{w}_{out}-\sum_{j=1}^{l_w} c_{wj}(\dot{w}_{out}-\dot{w}_j)-\sum_{j=1}^{l_w} k_{wj}(w_{out}-w_j)=-\sum_{j=1}^{l_w} T_{0j}+0.5T_{ecc}-T_{c\_out}-T_{s\_stiff}-T_{s\_damp},\\
&T_{s\_stif}=k_{ts}\left(w_{inp}-w_{out}\right),\\
&T_{s\_damp}=c_{ts}\left(\dot{w}_{inp}-\dot{w}_{out}\right),
\end{aligned}
\tag{50}
$$

In equations 6.2.1 and 6.2.2 the following notations are used:

m and I	shaft mass and moment of inertia about the rotation axis
x and y	coordinates of shaft center of gravity
w	shaft rotation angle
x_i, y_i and w_i	coordinates and rotation angle of i-th connected elements on input end
x_j, y_j and w_j	coordinates and rotation angle of the j-th connected elements on output end
c_{xi} and c_{xj}	damping coefficients (functions) of the i-th input and j-th output mechanical connections in x-direction
k_{xi} and k_{xj}	stiffness coefficients (functions) of the i-th input and j-th output mechanical connections in x-direction
c_{yi} and c_{yj}	damping coefficients (functions) of the i-th input and j-th output mechanical connections in y-direction
k_{yi} and k_{yj}	stiffness coefficients (functions) of the i-th input and j-th output mechanical connections in y-direction
c_{wi} and c_{wj}	torsional damping coefficients (functions) of the i-th input and j-th output mechanical connections in w direction
k_{wi} and k_{wj}	torsional stiffness coefficients (functions) of the i-th input and j-th output mechanical connections in w direction
F_{0xi} and F_{0xj}	preload forces of the i-th input and j-th output mechanical connections in x-direction

F_{0yi} and F_{0yj}	preload forces of the i-th input and j-th output mechanical connections in y-direction
T_{0i} and T_{0j}	preload torques of the i-th input and j-th output mechanical connections in w direction
n_x and l_x , n_y and l_y , nw and lw	number of mechanical connections on input and output ends in x, y and w directions, respectively
F_{x_ext} and F_{y_ext}	external forces in x and y-directions resulting from Coulomb friction and shaft unbalance
T_{ext}	external torque around rotation axis caused by Coulomb friction and unbalance

☞ **Note:** If the coordinate system of the **Shaft** element and local coordinate systems of connected elements and connections are not translated but there is an angle between their local x-directions (some of them are rotated to the global coordinate system), see Section 6.2.9.

Moments of inertia around the rotation axis I_{p_inp} and I_{p_out} are calculated from the geometric and material data the shaft pieces (where I_{p_inp}+I_{p_ou}=I_p).

Damping and stiffness coefficients (functions) and preload forces in Equation 49 are specified in the input dialogs of respective mechanical connections (refer to Chapter 21).

If there is an angle between local (cam) and global (shaft) coordinate systems then definition of mechanical connection stiffness, damping and preload is done in global coordinate system.

External forces F_{x_ext} , F_{y_ext} and torque T_{ext} consist of two components:

$$\begin{aligned} F_{x\_ext} &= F_{x\_ecc} - F_{frict}, \\ F_{y\_ext} &= F_{y\_ecc} - F_{frict}, \\ T_{ext} &= T_{ecc} - T_{frict}, \end{aligned} \tag{51}$$

where F_{frict} is the Coulomb friction force in x and y*-directions,* T_{frict} is the moment of Coulomb friction force around rotation axis, F_{x_ecc} and F_{y_ecc} are the centrifugal forces in x and y*-directions* resulting from shaft unbalance and T_{ecc} is the unbalance torque around rotation axis.

Centrifugal forces F_{x_ecc} and F_{y_ecc} are obtained from the equations:

$$\begin{aligned} F_{x\_ecc} &= me\left(\ddot{w}\sin(w+\alpha) + \dot{w}^2\cos(w+\alpha)\right), \\ F_{y\_ecc} &= me\left(-\ddot{w}\cos(w+\alpha) + \dot{w}^2\sin(w+\alpha)\right), \end{aligned} \tag{52}$$

where e is the eccentricity of shaft center of gravity and α is the initial angle between x axis and straight line connecting center of gravity with geometric center (coordinate origin). Input parameters e and α are depicted in Figure 47.

Unbalance torque $Tecc$ is calculated from the following equation:

$$T_{ecc} = e\left(\left(\sum_{con} F_y + F_{y\_ext}\right)\sin(w+\alpha) - \left(\sum_{con} F_x + F_{x\_ext}\right)\cos(w+\alpha)\right), \tag{53}$$

where $\sum_{con} F_x$ and $\sum_{con} F_y$ are the resultant forces from all mechanical connections in x and *y-directions,* respectively.

If shaft has no unbalance (e=0) and damping and stiffness coefficients of all mechanical connections are constant, Equation 49 is linear, otherwise non-linear. Variable connection implies that stiffness is a function of relative displacement and/or damping is a function of relative displacement or velocity.

If a **Shaft** element is attached to the input end of **Cam Profile** or **Rocker Arm (Cam)** and there is an angle between cam (local), mechanical connection (local) and shaft (global) coordinate systems, BOOST Hydsim will transform coordinates and velocities of the connected elements into local coordinate system of mechanical connection and calculate the resulting force. Components of this force will act on **Shaft** in local x and y direction.

6.2.8. Dynamics of Multi-body Systems

In BOOST Hydsim, each rigid or elastic mechanical element (e.g. **Mass, Shaft, Piston, Plunger, Needle, Valve Body,** etc.) has a predefined number of degrees of freedom (DOFs). For instance, **Rigid Shaft** element has three DOFs (translation along x and y axes and rotation about w axis), **Elastic Shaft** element - four DOFs (translation along x and y axes and rotation of both ends about w axis). **Mass** element has two DOFs (translation along x and y axes). Other mechanical elements have one DOF (translation along x axis) for a **Rigid body** case or two DOFs (translation along x axis) for an **Elastic body** case. For each DOF, mechanical connections to other elements can be specified. These are defined by linear/non-linear stiffness, damping, and preload force (torque). Along the x axis, hydraulic connections can be defined additionally for specific elements.

Equations of motion of a multi-body system can be derived by Newton's Second Law of Motion, d'Alembert's Principle or Lagrange's energy equation. In any case, this results in a system of ordinary differential equation of second order with external excitation:

$$[\mathbf{M}]\ddot{\mathbf{q}} + [\mathbf{C}]\dot{\mathbf{q}} + [\mathbf{K}]\mathbf{q} + \{\mathbf{R}(\mathbf{q}, \dot{\mathbf{q}}, \ddot{\mathbf{q}}, \mathbf{p})\} = \{\mathbf{F}(t)\} \tag{54}$$

where:

[M], **[C]** and **[K]** are the inertia, damping, and stiffness matrices, respectively

$\mathbf{q}$ is the vector of generalised coordinates (displacements and rotation angles)

$\{\mathbf{R}(\mathbf{q}, \dot{\mathbf{q}}, \ddot{\mathbf{q}}, \mathbf{p})\}$ is the vector of internal non-linear forces

$\mathbf{p}$ is the vector of pressures in hydraulic elements (e. g. volumes)

$\mathbf{F}(t)$ is the vector of external generalised forces

Vector of generalised coordinates $\mathbf{q}$ is defined as follows:

$$\mathbf{q}(t)=\{\mathbf{x},\mathbf{y},\varphi\} \tag{55}$$

where $\mathbf{x}$ and $\mathbf{y}$ are the displacement vectors in x and *y-directions,* respectively, and φ is the vector of rotation angles around w axis.

Pressure vector $\mathbf{p}$ contains the pressures in all volume elements which exert hydraulic forces on the connected mechanical bodies.

In BOOST Hydsim, matrix equation (Equation 54) is not explicitly used, mainly because of the diverse structure of the vector $\{\mathbf{R}(\mathbf{q},\dot{\mathbf{q}},\ddot{\mathbf{q}},\mathbf{p})\}$, which cannot be analytically formalised. BOOST Hydsim currently applies the fourth order Runge-Kutta-Gill scheme for the integration of ordinary second-order differential equations, which requires the reduction of Equation 54 to the first order system of a specific form. This method can well handle different types of nonlinearities occurring in the physical systems modelled by BOOST Hydsim.

In the general case, Equation 54 can be expanded into the following form:

$$\begin{gathered}
m_1\ddot{x}_1+\sum_{i=1}^{n_{x1}}c_{x1,i}(\dot{x}_1-\dot{x}_i)+\sum_{i=1}^{n_{x1}}k_{x1,i}(x_1-x_i)+R_{x1}(q,p,t)=\sum_{i=1}^{n_{x1}}F^0_{x1,i}+F^{ext}_{x1}(t),\\
\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\\
m_k\ddot{x}_k+\sum_{i=1}^{n_{xk}}c_{xk,i}(\dot{x}_k-\dot{x}_i)+\sum_{i=1}^{n_{xk}}k_{xk,i}(x_1-x_i)+R_{xk}(q,p,t)=\sum_{i=1}^{n_{xk}}F^0_{xk,i}+F^{ext}_{xk}(t),\\
m_1\ddot{y}_1+\sum_{i=1}^{n_{y1}}c_{y1,i}(\dot{x}_1-\dot{x}_i)+\sum_{i=1}^{n_{y1}}k_{y1,i}(x_1-x_i)+R_{y1}(q,p,t)=\sum_{i=1}^{n_{y1}}F^0_{y1,i}+F^{ext}_{y1}(t),\\
\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\\
m_l\ddot{y}_l+\sum_{i=1}^{n_{yl}}c_{yl,i}(\dot{x}_1-\dot{x}_i)+\sum_{i=1}^{n_{yl}}k_{yl,i}(x_1-x_i)+R_{yl}(q,p,t)=\sum_{i=1}^{n_{y1}}F^0_{yl,i}+F^{ext}_{yl}(t),\\
I_1\ddot{\varphi}_1+\sum_{i=1}^{n_{w1}}c_{w1,i}(\dot{\varphi}_1-\dot{\varphi}_i)+\sum_{i=1}^{n_{w1}}k_{w1,i}(\varphi_1-\varphi_i)+R_{w1}(q,p,t)=\sum_{i=1}^{n_{w1}}T^0_{w1,i}+T^{ext}_{w1}(t),\\
\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\\
I_s\ddot{\varphi}_s+\sum_{i=1}^{n_{ws}}c_{ws,i}(\dot{\varphi}_s-\dot{\varphi}_i)+\sum_{i=1}^{n_{ws}}k_{ws,i}(\varphi_s-\varphi_i)+R_{ws}(q,p,t)=\sum_{i=1}^{n_{ws}}T^0_{ws,i}+T^{ext}_{ws}(t),
\end{gathered} \tag{56}$$

where:

$x_1,\ldots,x_k$	displacements in *x-direction*
$y_1,\ldots,y_l$	displacements in *y-direction*
$\varphi_1,\ldots,\varphi_s$	rotation angles around *w* axis
$m_1,\ldots,m_k$	moving masses in *x-direction*

$m_1,...,m_l$	moving masses in *y-direction*
$I_1,...,I_s$	mass moments of inertia around rotation axis *w*
$c_{x1},...,c_{xk}$	damping coefficients (or functions) of the mechanical connections in *x-direction*
$c_{y1},...,c_{yl}$	damping coefficients (or functions) of the mechanical connections in *y-direction*
$c_{w1},...,c_{ws}$	torsional damping coefficients (or functions) of the mechanical connections in *w* direction
$k_{x1},...,k_{xk}$	stiffness coefficients (or functions) of the mechanical connections in *x-direction*
$k_{y1},...,k_{yl}$	stiffness coefficients (or functions) of the mechanical connections in *y-direction*
$k_{w1},...,k_{ws}$	torsional stiffness coefficients (or functions) of the mechanical connections in *w* direction
$R_{x1}(q,p,t),...,R_{xk}(q,p,t)$	internal forces in *x-direction*
$R_{y1}(q,p,t),...,R_{yl}(q,p,t)$	internal forces in *y-direction*
$R_{w1}(q,p,t),...,R_{ws}(q,p,t)$	internal torques about the rotation axis *w*
$F^0_{x1},...,F^0_{xk}$	preload forces of the mechanical connections in *x-direction*
$F^0_{y1},...,F^0_{yl}$	preload forces of the mechanical connections in *y-direction*
$T^0_{w1},...,T^0_{ws}$	preload torques of the mechanical connections in *w* direction
$F^{ext}_{x1}(t),...,F^{ext}_{xk}(t)$	external forces in *x-direction*
$F^{ext}_{y1}(t),...,F^{ext}_{yl}(t)$	external forces in *y-direction*
$T^{ext}_{w1}(t),...,T^{ext}_{ws}(t)$	external torques about rotation axis *w*
$n_{x1},...,n_{xk}$	numbers of mechanical connections in *x-direction*
$n_{y1},...,n_{yl}$	numbers of mechanical connections in *y-direction*
$n_{w1},...,n_{ws}$	numbers of mechanical connections in *w* direction

Stiffness, damping and preload of mechanical connections can be either constants or functions of generalised coordinates (relative displacement/angle or relative velocity):

$c_{x_i} = c_{x_i}(\Delta x_i, \Delta\dot{x}_i)$	$k_{x_i} = k_{x_i}(\Delta x_i)$	$F_{x_i} = F_{x_i}(\Delta x_i)$
$c_{y_i} = c_{y_i}(\Delta y_i, \Delta\dot{y}_i)$	$k_{y_i} = k_{y_i}(\Delta y_i)$	$F_{y_i} = F_{y_i}(\Delta y_i)$
$c_{w_i} = c_{w_i}(\Delta\varphi_i, \Delta\dot{\varphi}_i)$	$k_{w_i} = k_{w_i}(\Delta\varphi_i)$	$T_{w_i} = T_{w_i}(\Delta\varphi_i)$

Internal forces depend on the type of the element and its connections. For **Piston/Plunger** elements, internal forces could be hydraulic forces caused by pressure acting on input and output area (if hydraulic connections are defined).

For **Shaft** element, internal forces could be centrifugal forces from the eccentricity. An example of external force is friction force. It can be defined for all mechanical elements. Magnetic force acting on **Solenoid Valve Body** can be internal or external force depending on the type of **Solenoid** element used. External forces can be specified as arbitrary time functions using the **Modify Parameter** option. Note that for certain elements (e.g. **Needle, SID Piston, Solenoid Armature)** the forces acting at input and output ends are different, therefore the wrong definition of input/output connections may lead to false results.

For every type of mechanical elements, Equation 54 can be expanded into the following form:

- Shaft (three DOF)

$$
\begin{aligned}
& m\ddot{x}+\sum_{i=1}^{n_x}\left\{c_{xi}\left(\dot{x}-\dot{x}_i\right)+k_{xi}\left(x-x_i\right)-F_{xi}\right\}=\sum_{j=1}^{m_x}F_{xj}^{ext}(t)\ , \\
& m\ddot{y}+\sum_{i=1}^{n_y}\left\{c_{yi}\left(\dot{y}-\dot{y}_i\right)+k_{yi}\left(y-y_i\right)-F_{yi}\right\}=\sum_{j=1}^{m_y}F_{yj}^{ext}(t)\ , \\
& I\ddot{\varphi}+\sum_{i=1}^{n_w}\left\{c_{wi}\left(\dot{\varphi}-\dot{\varphi}_i\right)+k_{wi}\left(\varphi-\varphi_i\right)-T_{wi}\right\}=\sum_{j=1}^{m_w}T_{wj}^{ext}(t)\ ,
\end{aligned}
\tag{57}
$$

where:

m	moving mass
I	shaft moment of inertia around rotation axis w
c_{x_i}, c_{y_i}, and c_{w_i}	damping coefficients (or functions) of i-th mechanical connection in the respective directions.
k_{x_i}, k_{y_i}, and k_{w_i}	stiffness coefficients (or functions) of i-th connections in the respective direction.
F_{x_i} and F_{y_i}	preload forces of the i-th connection in the directions x and y
T_{w_i}	preload torque of the i-th connection about rotation axis w
$F_{xj}^{ext}(t)$ and $F_{yj}^{ext}(t)$	external forces acting on the shaft in x and y-direction
$T_{w_i}^{ext}(t)$	external torque around rotation axis w
x, y, and φ	generalised coordinates
nx, ny and nw	number of mechanical connections in the respective directions
mx, my, and mw	number of external forces (torques) in the respective directions

External forces for **Shaft** element can be centrifugal forces from eccentricity of gravity center, and friction forces/torque if specified in the input dialog.

- Mass (two DOF)

$$m\ddot{x}+\sum_{i=1}^{n_x}\left\{c_{xi}\left(\dot{x}-\dot{x}_i\right)+k_{xi}\left(x-x_i\right)-F_{xi}\right\}=\sum_{j=1}^{m_x}F_{xj}^{ext}(t)\ ,$$
$$m\ddot{y}+\sum_{i=1}^{n_y}\left\{c_{yi}\left(\dot{y}-\dot{y}_i\right)+k_{yi}\left(y-y_i\right)-F_{yi}\right\}=\sum_{j=1}^{m_y}F_{yj}^{ext}(t)\ , \tag{58}$$

6.2.9. Equations of Motion (Local/global coordinate system)

Force $F_{mc\_x\_ij}$ in the mechanical connections in *x-direction* between i-th and j-th element is calculated by the following equation:

$$F_{mc\_x\_ij}=c_{xij}\left(\dot{x}_i\cos\alpha_{mc\_in}-\dot{x}_j\cos\alpha_{mc\_out}\right)+k_{xij}\left(x_i\cos\alpha_{mc\_in}-x_j\cos\alpha_{mc\_out}\right)+F_{mc\_x\_0\_ij}, \tag{59}$$

Force $F_{mc\_y\_ij}$ in the mechanical connections in *y-direction* between i-th and j-th element is calculated by the following equation:

$$F_{mc\_y\_ij}=c_{yij}\left(\dot{y}_i\cos\alpha_{mc\_in}-\dot{y}_j\cos\alpha_{mc\_out}\right)+k_{yij}\left(y_i\cos\alpha_{mc\_in}-y_j\cos\alpha_{mc\_out}\right)+F_{mc\_y\_0\_ij}, \tag{60}$$

where angles are:

$$\alpha_{mc\_in}=\alpha_{mc\_ij}-\alpha_i\quad,\quad\alpha_{mc\_out}=\alpha_j-\alpha_{mc\_ij}$$

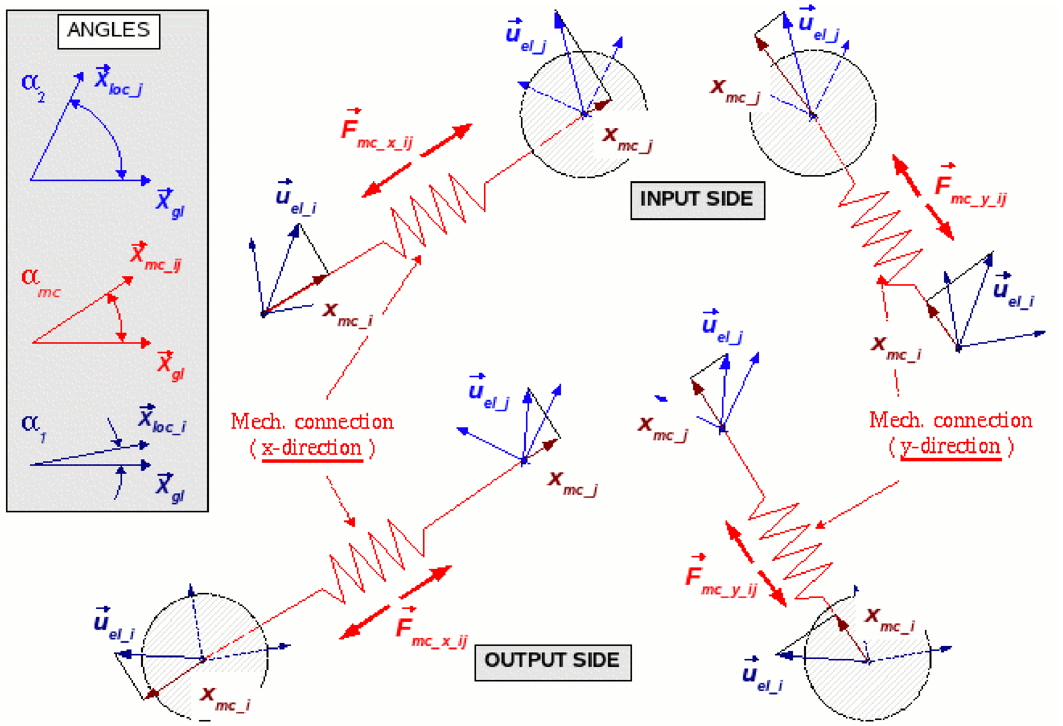


Figure 51: Forces in Mechanical Connections

The motion of *i-th* **Rigid Body** is governed by the following equations:

$$
\begin{aligned}
m\ddot{x}_i + F_{mc\_x\_i} &= \sum_{j=1}^{m_x} F_{xij}^{ext}(t) \ , \\
m\ddot{y}_i + F_{mc\_y\_i} &= \sum_{j=1}^{m_y} F_{yij}^{ext}(t) \ ,
\end{aligned}
\tag{61}
$$

where:

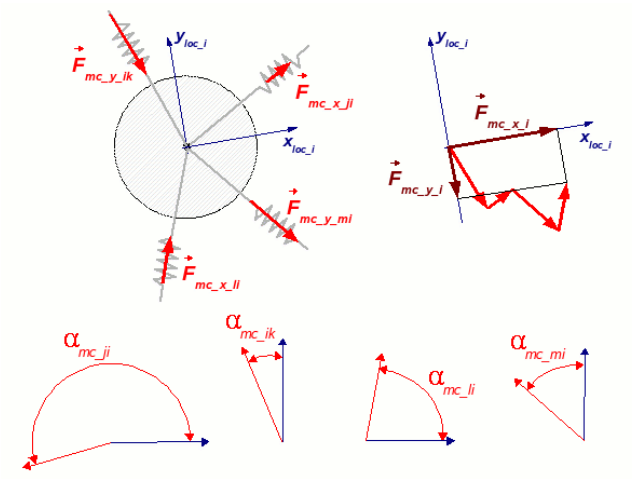
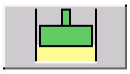
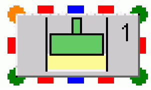


Figure 52: Forces from Mechanical Connections on Rigid Body

7. PISTON

7.1. Standard Piston

Element Name:	**Standard Piston**	
Element Icon:		
Definition:	This element serves to define a standard (hydraulic) Piston.	
Connecting pins:	standard pins: 8 (6 mechanical and 2 hydraulic) special pins: 3 wire pins: 1	

Note: The rigid **Piston** element has only one degree-of-freedom (translation in x direction). Elastic **Piston** has two DOFs: translation along x-axis for input and output end, separately.

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **Piston** may only have one hydraulic connection on each end (input and output)

Note: Mechanical connections can be defined only in local x-direction of connection.

Note: For **Standard Piston** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters such as coordinates, velocities and accelerations are also calculated in the local coordinate system.

Note: Local x axis of **Standard Piston** may be **translated** and **rotated** in the global coordinate system. Rotation is defined through parameter: Direction of Element motion.

Mechanical model of **Standard Piston** with possible connections is shown Figure 53.

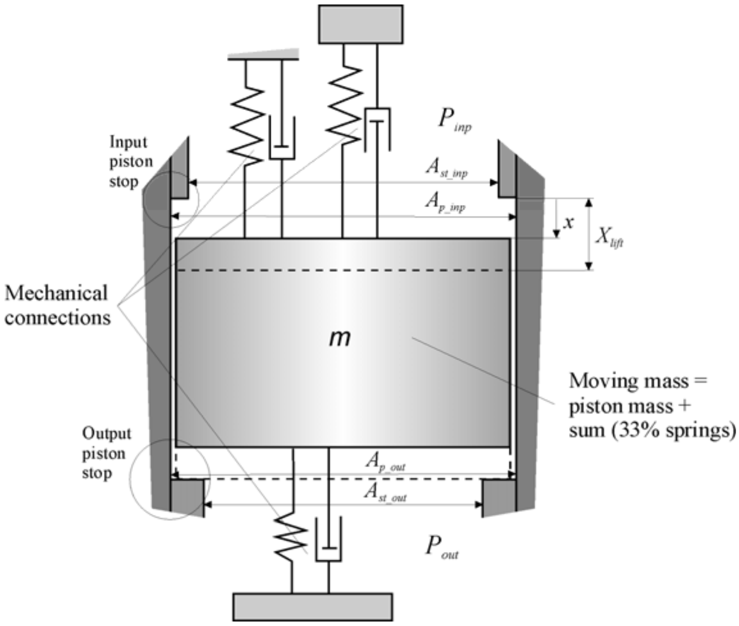


Figure 53: Mechanical Model of Piston

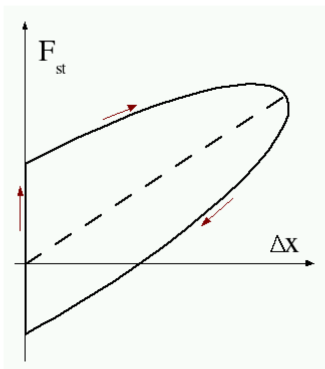
7.1.1. Input Parameters

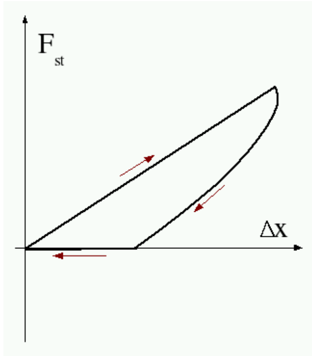
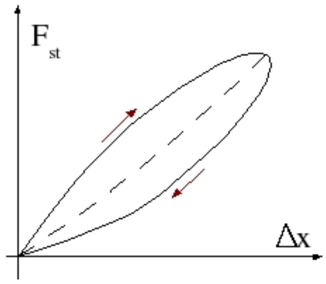
Description of input data for the **Standard Piston**:

Element name:	The name of the element is specified as default.
Rigid Body	**Rigid Piston** is a rigid body with 1 DOF, i.e. without elasticity between the input and output end. Coordinates and velocities on input and output end of **Piston** element are same.
Moving mass	Specify moving mass (m). It is equal to the piston mass plus 33% of the total mass of the attached springs (mechanical connections).
Coulomb friction force	Specify the Coulomb friction force in local x-direction.
Elastic Body	**Elastic Piston** implies an elastic body with the axial elasticity between the input and output end. Stiffness and damping characteristics of **Elastic Piston** are calculated according to the geometric and material data.
Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the elastic piston (refer to 6.2.2).
Piston stroke (max. lift)	Specify distance between piston input and output stops. Piston stroke (maximum lift) corresponds to X_{lift} in Figure 53. **Note:** If button is not active, piston output stop data is faded out.
CROSS-SECTION (UNDER PRESSURE)	
Cross-section Area	Specify the cross-sectional areas of piston.
Piston area at input end	Specify the cross-sectional area of piston at input end A_{p_inp} (refer to Figure 53).

Piston area at output end	Specify the cross-sectional area of piston at output end A_{p_out} (refer to Figure 53).
Cross-section Diameter	Specify the diameters of piston.
Piston diameter at input end	Specify the diameter of piston at input end d_{p_inp}.
Piston diameter at output end	Specify the diameter of piston at output end d_{p_out}.
STOP CROSS-SECTION (UNDER PRESSURE)	
Stop Area	Specify the stop cross-sectional areas of piston.
Piston stop area at input end	Specify the stop cross-sectional area of piston at input end A_{st_inp} (refer to Figure 53).
Piston stop area at output end	Specify the cross-sectional area of piston at output end A_{st_out} (refer to Figure 53).
Stop Diameter	Specify the stop diameters of piston.
Piston stop diameter at input end	Specify the stop diameter of piston at input end d_{st_inp} .
Piston stop diameter at output end	Specify the diameter of piston at output end d_{st_out} .

If the **Piston** input/output end is attached to **Volume** or **Pressure Boundary** element, the piston input/output area will be under pressure. This will produce hydraulic connection forces that will enter the dynamical equation of motion of piston as additional forces (refer to Equation of Motion (62)). Furthermore, piston velocity multiplied by piston area will produce flow rate which will enter the continuity equation of attached **Volume** element (refer to Sections 8.1.6 and 8.2.6).

STOP MODEL	
Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations 64 and 67).  **Figure 54: Linear Stop Model**

Stop model is pseudo-linear	This model is derived from the linear model by introducing an additional switching function (see equations 65 and 68). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact.  **Figure 55: Pseudo-linear stop model**
Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model[1] model is used.  **Figure 56: Non-linear stop model**
PISTON STOP DATA	
(Linear or Pseudo-linear Stop model)	
Piston stop stiffness at input end	Specify the piston stop stiffness on input end (k_{in_st}). Stiffness is active when piston lift $x < 0$.
Piston stop stiffness at output end	Specify the piston stop stiffness on output end (k_{out_st}). Stiffness is active when piston lift $x > X_{lift}$.
Piston stop damping at input end	Specify the piston stop damping on input end (c_{in_st}). Damping is active when piston lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Piston stop damping at output end	Specify the piston stop damping on output end (c_{out_st}).. Damping is active when piston lift $x > X_{lift}$. (and for pseudo-linear stop model when $\dot{x} \leq 0$).

[1] K. H. Hunt, F. R. E. Crossley, *Coefficient of Restitution Ineterpreted as Damping in Vibroimpact*, ACME Journal of Applied Mechanics, 1975.

(Nonlinear Stop model)	
Piston stop linear stiffness at input end	Specify the piston stop linear stiffness on input end (k_{in_st}). Stiffness is active when piston lift $x < 0$.
Piston stop linear stiffness at output end	Specify the piston stop linear stiffness on output end (k_{out_st}). Stiffness is active when piston lift $x > X_{lift}$.
Piston stop Stiffness factor at input end	Specify the piston stop stiffness factor on input end (n_{in_st}).
Piston stop Stiffness factor at output end	Specify the piston stop stiffness factor on output end (n_{out_st}).
Piston stop Elastic impact coef. at input end	Specify the piston stop elastic impact coefficient on input end (α_{in_st}).
Piston stop Elastic impact coef. at output end	Specify the piston stop elastic impact coefficient on output end (α_{out_st}).
The stiffness and damping parameters of the piston stops may be determined approximately: they are necessary to prevent numerical discontinuities. As a rule, the stiffness of piston stop should be at least several orders of magnitude higher than the corresponding stiffness of the mechanical connection (spring).	
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$,
y(e2)	where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the
z(e3)	global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}).	

7.1.2. Elastic Piston Properties

Elastic piston option allows to calculate the deformation of the piston as a result of the forces acting on its input and output end. Stiffness k_{ps} and damping c_{ps} of the virtual spring and damper represent the axial elasticity between the input and output end as shown in Figure 57.

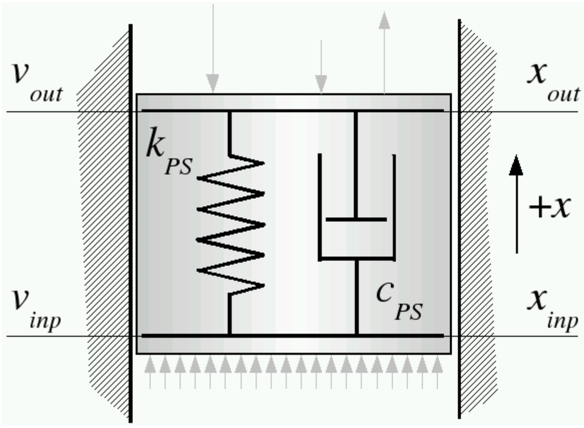
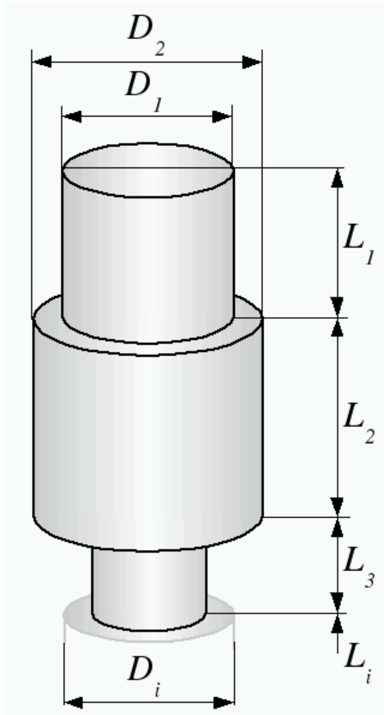


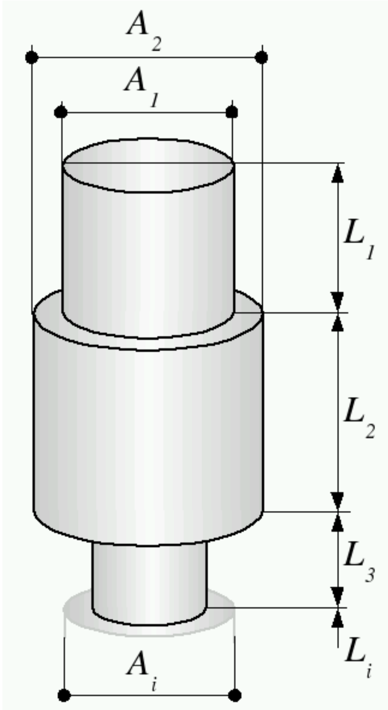
Figure 57: Mechanical Model of Elastic Piston

For the internal calculation of the axial stiffness k_{ps} and damping c_{ps}, full geometry and material data of the piston have to be specified in the dialog **Elastic Piston**.

Description of input data:

Material Data	
Solid Properties …	Press this bar to define the local solid properties (refer to Section 8.2.2). Global solid properties are active by default.
Equivalent viscous damping ratio	Specify equivalent viscous damping ratio (r_C). Default value is set to critical damping.
Geometric Data	
Cross-section type:	
Circular	Geometry and input data of the elastic piston of circular cross-section is shown in Figure 58.
Cross-section table	Specify diameters D_i and lengths L_i of all piston sections.

	 **Figure 58: Piston with Circular Cross-sections** Axial piston stiffness k_{ps} is calculated form the stiffness of each piston section k_i: $$\frac{1}{k_{ps}} = \sum_i \frac{1}{k_i} \quad \text{where} \quad k_i = \frac{EA_i}{L_i}$$ Piston cross-section area is given by: $$A_i = \frac{\pi D_i^2}{4}$$ Piston damping c_{ps} is calculated from the critical damping c_{p_crit}: $$c_{p\_crit} = 2.\sqrt{mk_{ps}}$$ $$c_{ps} = r_c c_{p\_crit}$$
Other	Piston model with other cross-sections is shown in Figure 59.
Cross-section table	Specify areas A_i and lengths L_i of all piston sections.

	 **Figure 59: Piston with Other Cross-sections** Axial piston stiffness k_{ps} is calculated form the stiffness of each piston section k_i: $\frac{1}{k_{ps}} = \sum_i \frac{1}{k_i}$ where $k_i = \frac{EA_i}{L_i}$ Piston damping c_{ps} is calculated from the critical damping c_{p_crit}: $c_{p\_crit} = 2\sqrt{mk_{ps}}$ $c_{ps} = r_c c_{p\_crit}$
Additional mass (spring)	Specify additional mass of piston (rigid part). Additional mass may be used to define the mass of attached springs (33% of the total spring mass).
Rigid mass at input end	Specify additional rigid mass at input end m_{ad_inp}.
Rigid mass at output end	Specify additional rigid mass at input end m_{ad_out}.
Coulomb friction force	
Friction force at input end	Specify additional Coulomb friction force on the input side of piston element F_{C_inp}.
Friction force at output end	Specify additional Coulomb friction force on the output side of piston element F_{C_out}.

7.1.3. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial conditions:

X-coordinate	
input x	Specify the initial position of piston input end in local x-direction.
output x	Specify the initial position of piston output end in local x-direction.
	Note: If **Rigid Body** option is active, then initial coordinates at input and output end must be the same.
Velocity in x-direction	
input v	Specify the initial velocity of piston input end in local x-direction.
output v	Specify the initial velocity of piston output end in local x-direction.
	Note: If **Rigid Body** option is active, then initial velocities at input and output end must be the same.

All initial values that are not specified under initials are set to 0.

7.1.4. Modifiable Parameters

Path: **Element | Initial Conditions**

To activate modification, the option button to the left of modifiable parameter name must be checked. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

Moving mass	SI Unit:	kg
Cross-sectional area at input end	SI Unit:	m^2
Cross-sectional area at output end	SI Unit:	m^2
Coulomb friction force	SI Unit:	N
Piston stroke (maximum lift)	SI Unit:	m
Piston diameter at input end	SI Unit:	m
Piston diameter at output end	SI Unit:	m

7.1.5. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter name has to be checked.

Description of output parameters:

x-coordinate (input)	Local x-coordinate of piston input end.
velocity in x-direction (input)	Velocity of piston input end in x-direction.
acceleration in x-direction (input)	Acceleration of piston input end in x-direction.
x-coordinate (output)	Local x-coordinate of piston output end.
velocity in x-direction (output)	Velocity of piston output end in x-direction.
acceleration in x-direction (output)	Acceleration of piston output end in x-direction.

> ☞ **Note:** If **Rigid Body** option is active, then initial coordinates and velocities at input and output end must be the same.

Forces are calculated in local x-direction:	
resultant force	Resultant force of piston is the sum of all connection forces and external forces acting on it. $F_R = m\ddot{x}$
hydraulic force	Hydraulic force is the sum of pressure-induced forces on input and output ends.
mechanical force	Mechanical force is the sum of the preload forces, stiffness and damping forces, Coulomb friction force and forces from the piston stops.
shear force from leakage	Shear force is calculated in **Annular gap** element, which is connected to piston element via special connection. (refer to Section **13.1.5.1**, Equation **236**)

7.1.6. Equation of Motion of Rigid Piston

The motion of **Rigid Piston** in local coordinate system is governed by the following equation (62):

$$
\begin{aligned}
m\ddot{x} \quad & +\sum_{i=1}^{n} c_i\left[\dot{x}\cos(\alpha-\alpha_{mc\_i})-\dot{x}_i\cos(\alpha_{mc\_i}-\alpha_i)-\dot{y}_i\sin(\alpha_{mc\_i}-\alpha_i)\right]\cos(\alpha-\alpha_{mc\_i}) \\
& +\sum_{i=1}^{n} k_i\left[x\cos(\alpha-\alpha_{mc\_i})-x_i\cos(\alpha_{mc\_i}-\alpha_i)-y_i\sin(\alpha_{mc\_i}-\alpha_i)\right]\cos(\alpha-\alpha_{mc\_i}) \\
& -\sum_{j=1}^{m} c_j\left[\dot{x}\cos(\alpha_{mcj}-\alpha)-\dot{x}_j\cos(\alpha_j-\alpha_{mc\_j})+\dot{y}_j\sin(\alpha_j-\alpha_{mc\_j})\right]\cos(\alpha_{mcj}-\alpha) \\
& -\sum_{j=1}^{m} k_j\left[x\cos(\alpha_{mcj}-\alpha)-x_j\cos(\alpha_j-\alpha_{mc\_j})+y_j\sin(\alpha_j-\alpha_{mc\_j})\right]\cos(\alpha_{mcj}-\alpha) \\
-\sum_{i=1}^{n} F_{0i}\cos(\alpha-\alpha_{mc\_i}) & +\sum_{j=1}^{m} F_{0j}\cos(\alpha_{mc\_j}-\alpha)=F_{hyd}-F_{frict}-F_{shear}-F_{in\_st}-F_{out\_st}
\end{aligned}
\tag{62}
$$

where:

m	Piston mass
x	local x-coordinate of piston
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends
Forces on right side of equation are defined in local x-direction:	
F_{hyd}	hydraulic force in local x-direction
F_{in_st}, F_{out_st}	additional forces from input and output stops

F_{frict}	Coulomb friction force
F_{shear}	viscous friction force due to leakage (if any)

Depending on the Piston position, hydraulic force F_{hyd} is calculated from the equation:

$$\begin{aligned} \forall x \le 0 \quad &: \quad F_{hyd} = p_{inp} A_{st\_inp} - p_{out} A_{p\_out} \,, \\ \forall 0 < x < X_{lift} \quad &: \quad F_{hyd} = p_{inp} A_{p\_inp} - p_{out} A_{p\_out} \\ \forall x \ge X_{lift} \quad &: \quad F_{hyd} = p_{inp} A_{p\_inp} - p_{out} A_{st\_out} \end{aligned} \tag{63}$$

where p_{inp} and p_{out} are the input and output pressures, A_{p_inp} and A_{p_out} are the input and output cross-sectional areas of piston, A_{st_inp} and A_{st_out} are the input and output cross-sectional areas of piston seat and stop, respectively.

Force from input stop F_{in_st} is defined as follows:

Linear stop model

$$\begin{aligned} &\forall\, x < 0 : F_{in\_st} = c_{in\_st}\dot{x} + k_{in\_st} x \,, \\ &\forall\, x \ge 0 : F_{in\_st} = 0, \end{aligned} \tag{64}$$

Pseudo-linear stop model

$$\begin{aligned} &\forall x < 0 \\ &\qquad \forall \dot{x} < 0 \quad : \quad F_{in\_st} = k_{in\_st} x \,, \\ &\qquad \forall \dot{x} \ge 0 \quad : \quad F_{in\_st} = c_{in\_st}\dot{x} + k_{in\_st} x \,, \\ &\qquad \forall F_{in\_st} > 0 \quad : \quad F_{in\_st} = 0 \,, \\ &\forall x \ge 0 \\ &\qquad\qquad : \quad F_{in\_st} = 0 \,, \end{aligned} \tag{65}$$

where c_{in_st} and k_{in_st} are the damping and stiffness coefficients of the input stop.

Non-linear stop models

$$\begin{aligned} &\forall\, x < 0 : F_{in\_st} = k_{in\_st} (\text{-}\,\mathrm{x})^{\mathrm{n_{in\_st}}} \left(1 - \frac{3}{2}\alpha_{in\_st}\dot{x}\right), \\ &\forall\, x \ge 0 : F_{in\_st} = 0, \end{aligned} \tag{66}$$

where n_{in_st} is the stiffness factor and α_{in_st} is the elastic impact coefficient of the input stop.

Similarly, force from output stop F_{out_st} is given by:

Linear stop model

$$\begin{aligned} &\forall\, x \le X_{lift} : F_{out\_st} = 0 \,, \\ &\forall\, x > X_{lift} : F_{out\_st} = c_{out\_st}\dot{x} + k_{out\_st}\left(x - X_{lift}\right), \end{aligned} \tag{67}$$

Pseudo-linear stop model

$$
\begin{aligned}
&\forall x \le X_{lift} \\
&\qquad\qquad : \qquad F_{out\_st} = 0\,, \\
&\forall x > X_{lift} \\
&\qquad \forall \dot{x} > 0 \quad : \quad F_{out\_st} = k_{out\_st}\left(x - X_{lift}\right), \\
&\qquad \forall \dot{x} \le 0 \quad : \quad F_{out\_st} = c_{out\_st}\dot{x} + k_{out\_st}\left(x - X_{lift}\right), \\
&\qquad \forall \mathrm{F_{out\_st}} < 0 \quad : \quad F_{out\_st} = 0\,,
\end{aligned}
\tag{68}
$$

Non-linear stop model

$$
\begin{aligned}
&\forall\ x \le X_{lift} : F_{out\_st} = 0, \\
&\forall\ x > X_{lift} : F_{out\_st} = -k_{out\_st}\left(\mathrm{x - X_{lift}}\right)^{\mathrm{n_{out\_st}}}\left(1 + \frac{3}{2}\alpha_{out\_st}\dot{x}\right),
\end{aligned}
\tag{69}
$$

where n_{out_st} is the stiffness factor and α_{out_st} is the elastic impact coefficient of the output stop.

where c_{out_st} and k_{out_st} are the damping and stiffness coefficients of the output stop and X_{lift} is the piston stroke (distance between input and output stops).

Damping and stiffness coefficients of piston stops have to be specified directly in the input dialog of the **Standard Piston.**

Viscous friction (shear) force F_{shear} is included only if a **Leakage | Annular gap** element is attached (by special connection) to **Piston. Piston** element in this case must have a shape of a cylinder. Force F_{shear} is calculated from the Equation 236 (refer to the description of **Leakage** element in Chapter 13).

☞ **Note:** Viscous friction force depends on piston velocity. Without this force and with constant damping and stiffness coefficients of mechanical connections only, Equation (62) is linear. It becomes nonlinear if at least one mechanical connection is variable (stiffness is a function of relative displacement and damping is a function of relative displacement or relative velocity) or if viscous friction force exists.

7.1.7. Equation of Motion of Elastic Piston

Motion of **Piston with Elastic Body** in local coordinate system is governed by the following equations:

$$
\begin{aligned}
&\left(m_{inp}+m_{ad\_inp}\right)\ddot{x}_{inp} \\
&+\sum_{i=1}^{n} c_i\left[\dot{x}_{inp}\cos\left(\alpha-\alpha_{mc\_i}\right)-\dot{x}_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-\dot{y}_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&+\sum_{i=1}^{n} k_i\left[x_{inp}\cos\left(\alpha-\alpha_{mc\_i}\right)-x_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-y_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&-\sum_{i=1}^{n} F_{0i}\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&=F_{hyd\_inp}-F_{C\_inp}-0.5F_{shear}-F_{in\_st}+F_{stiff}+F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
&\left(m_{out}+m_{ad\_out}\right)\ddot{x}_{out} \\
&-\sum_{j=1}^{m} c_j\left[\dot{x}_{out}\cos\left(\alpha_{mcj}-\alpha\right)-\dot{x}_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+\dot{y}_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&-\sum_{j=1}^{m} k_j\left[x_{out}\cos\left(\alpha_{mcj}-\alpha\right)-x_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+y_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&+\sum_{j=1}^{m} F_{0j}\cos\left(\alpha_{mc\_j}-\alpha\right) \\
&=F_{hyd\_out}-F_{C\_out}-0.5F_{shear}-F_{out\_st}-F_{stiff}-F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
F_{stiff}&=k_{ps}\left(x_{inp}-x_{out}\right), \\
F_{damp}&=c_{ps}\left(\dot{x}_{inp}-\dot{x}_{out}\right)
\end{aligned}
$$

(70)

Masses m_{inp} and m_{out} are calculated according to the position of piston gravity centre ($m_{inp}+m_{out}=m$).

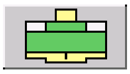
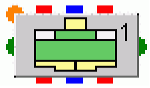
Depending on the piston end coordinates, hydraulic forces F_{hyd_inp} and F_{hyd_out} are calculated by:

$$
\begin{aligned}
\forall\, x_{inp}\le 0: &\quad F_{hyd\_inp}=p_{inp}A_{st\_inp}, \\
\forall\, x_{inp}> 0: &\quad F_{hyd\_inp}=p_{inp}A_{p\_inp}, \\
\forall\, x_{out}\le X_{lift}: &\quad F_{hyd\_out}=-p_{out}A_{p\_out}, \\
\forall\, x_{out}\ge X_{lift}: &\quad F_{hyd\_out}=-p_{out}A_{st\_out}
\end{aligned}
\qquad (71)
$$

where p_{inp} and p_{out} are the input and output pressures, A_{p_inp} and A_{p_out} are the input and output cross-sectional areas of the piston and A_{st_inp} and A_{st_out} are the input and output cross-sectional areas of piston seat and stop, respectively.

Note: Viscous friction (shear) force F_{shear} is split into two equal parts on input and output end of piston.

7.2. Split-Injection Piston

Element Name:	**Split-Injection Device (SID) Piston**	
Element Icon:		
Definition:	This element serves to define a Split-Injection-Device Piston. SID Piston is used for rate shaping of fuel injection (e.g. for pilot injection in some unit injectors).	
Connecting pins:	standard pins: 6 (4 mechanical and 2 hydraulic) special pins: 2 wire pins: 1	

Note: Currently the Variable-step solver does not support this element.

Note: **SID Piston** element has only one degree-of-freedom (translation in x direction).

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **SID Piston** may have only one hydraulic connection on each end (input and output)

Note: Mechanical connections can be defined only in local x-direction of connection.

Note: For **SID Piston** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: Local x axis of **SID Piston** may be **translated** and **rotated** in global coordinate system. Rotation is defined through definition of parameter: Direction of Element motion.

Note: Hydraulic and mechanical connections to SID Piston are **irreversible.** Each input connection (arrow pointing to the icon of SID piston) is treated as a connection to piston stop, and each output connection (arrow going out of the piston icon) as a connection to piston seat. Refer to Figure 60 for more

information.

The mechanical model of the SID Piston is shown in Figure 60. It has a mechanical connection (piston spring) on input side. Mechanical connection on output side is also formally possible, but not typical in practical applications.

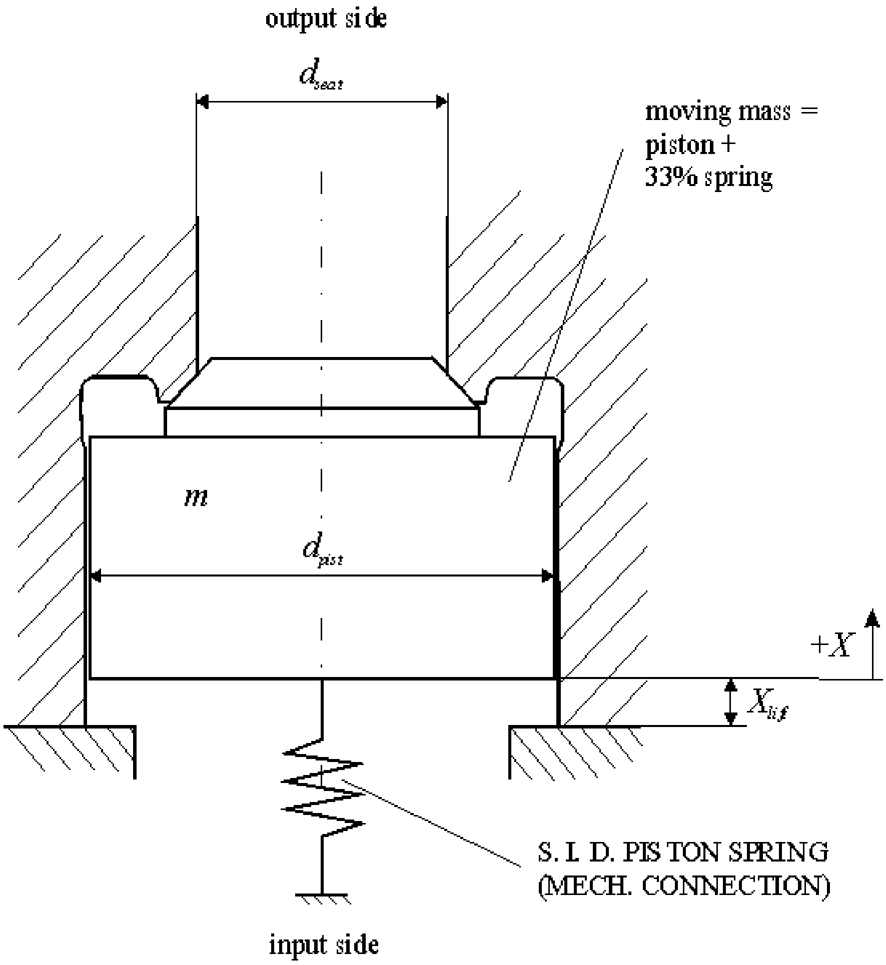


Figure 60: Mechanical Model of SID-Piston

7.2.1. Input Parameters

Description of input data for the **Split-Injection (SID) Piston**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Chapter 8).
Moving mass	Specify moving mass *m*. It is equal to the piston mass plus 33% of the total mass of the attached springs (mechanical connections).
Piston diameter	Specify diameter d_{pist} (refer to Figure 60).
Piston seat diameter	Specify diameter d_{seat} (refer to Figure 60).
From piston diameter and piston seat diameter areas on input and output side of piston are calculated. These areas are necessary for calculation of hydraulic connection forces.	

Coulomb friction force	Specify the Coulomb friction force in local x-direction.
Piston stroke (max. lift)	Specify distance between piston input and output stops. Piston stroke (maximum lift) corresponds to X_{lift} on Figure 60. **Note:** If button is not active, piston input stop data is faded out.
SQUEEZING FLUID AT PISTON CLOSING	
Damping coefficient	Specify coefficient of a velocity-proportional damping of squeezing fluid at piston closing (active for positive piston velocity $\dot{x} > 0$).
Piston seat/stop data	
(Linear or Pseudo-linear Seat/Stop model)	
Piston seat stiffness	Specify the piston seat stiffness (k_{in_st}) (output side). Stiffness is active when piston lift $x > 0$.
Piston stop stiffness	Specify the piston stop stiffness (k_{out_st}) (input side). Stiffness is active when piston lift $x < -X_{lift}$.
Piston seat damping	Specify the piston seat damping (c_{in_st}) (output side). Damping is active when piston lift $x > 0$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
Piston stop damping	Specify the piston stop damping (c_{out_st}) (input side). Damping is active when piston lift $x < -X_{lif}$. (and for pseudo-linear stop model when $\dot{x} \geq 0$).
(Nonlinear Seat/Stop model)	
Piston seat linear stiffness	Specify the piston seat linear stiffness (k_{in_st}) (output side). Stiffness is active when piston lift $x > 0$.
Piston stop linear stiffness	Specify the piston stop linear stiffness (k_{out_st}) (input side). Stiffness is active when piston lift $x < -X_{lift}$.
Piston seat Stiffness factor	Specify the piston seat stiffness factor (n_{in_st}).
Piston stop Stiffness factor	Specify the piston stop stiffness factor (n_{out_st}).
Piston seat Elastic impact coef.	Specify the piston seat elastic impact coefficient (α_{in_st}).
Piston stop Elastic impact coef.	Specify the piston stop elastic impact coefficient (α_{out_st}).
SEAT/STOP MODEL	
Seat/Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations 64 and 67) (refer to Figure 54).
Seat/Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function (see equations 65 and 68). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact (refer to Figure 55).

Stop model is non-linear	Linear model commonly used to describe the interaction with stiff environments sometime cannot be applied to soft materials, where viscous effects are relevant. For better physical consistency in describing this behavior it is taken non-linear model well known as Hunt-Crossley model.
DIRECTION OF ELEMENT MOTION	
x(e1) **y(e2)** **z(e3)**	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}).	

7.2.2. Initial Conditions

Initial Conditions for **Split Injection (SID) Piston** are identical to the Initial Conditions of **Standard Piston** (refer to section 7.1.3).

7.2.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be checked. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameter:

Moving mass	SI Unit:	kg
Damping of squeezing fluid at piston closing	SI Unit:	Ns/m
Maximum lift of piston	SI Unit:	m

7.2.4. Output Parameters

Path: **Elements | Store Results**

To activate output parameter, the option button in front of output parameter must be checked.

Description of output parameters:

x-coordinate (lift)	Local x-coordinate of **SID Piston.**
velocity in x-direction	Velocity in local x-direction of **SID Piston.**
acceleration in x-direction	Acceleration in local x-direction of **SID Piston.**
Forces are calculated in local x-direction:	
resultant force	Resultant force of piston element is sum of all connections and external forces acting on it: $F_R = m\ddot{x}$ Refer to Section 7.2.5
hydraulic force	Hydraulic force is result of pressure activity on input and output piston areas. Refer to Section 7.2.5, Equation 73
pressure drop at seat	Δp_{seat} is the pressure drop at piston seat due to throttling (at opening) Refer to Section 7.2.5, Equation 74
mechanical force	Mechanical force is sum of all preload forces, stiffness and damping forces, Coulomb friction forces and forces from stops. Refer to Section 7.2.5

Resultant hydraulic force is calculated from Equation 73 given in Section 7.2.5.

7.2.5. Equation of Motion

Motion of **SID Piston** in local coordinate system is governed by the following equation:

$$
\begin{aligned}
& m\ddot{x} \\
& + \sum_{i=1}^{n} c_i \left[\dot{x}\cos\left(\alpha - \alpha_{mc\_i}\right) - \dot{x}_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - \dot{y}_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
& + \sum_{i=1}^{n} k_i \left[x\cos\left(\alpha - \alpha_{mc\_i}\right) - x_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - y_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
& - \sum_{j=1}^{m} c_j \left[\dot{x}\cos\left(\alpha_{mcj} - \alpha\right) - \dot{x}_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + \dot{y}_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
& - \sum_{j=1}^{m} k_j \left[x\cos\left(\alpha_{mcj} - \alpha\right) - x_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + y_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
& - \sum_{i=1}^{n} F_{0i} \cos\left(\alpha - \alpha_{mc\_i}\right) + \sum_{j=1}^{m} F_{0j} \cos\left(\alpha_{mc\_j} - \alpha\right) \\
& = F_{hyd} - F_{frict} - F_{shear} - F_{damp} - F_{in\_st} - F_{out\_st}
\end{aligned}
\tag{72}
$$

where:

m	Piston mass
x	local x-coordinate of piston
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends
Forces on right side of equation are defined in local x-direction:	
F_{hyd}	resultant hydraulic force
F_{damp}	damping force from squeezing fluid at piston closing
F_{in_st} and F_{out_st}	additional forces from input and output stops
F_{frict}	Coulomb friction force
F_{shear}	viscous friction force due to leakage (if any)

Depending on the SID Piston position, hydraulic force $Fhyd$ is calculated from the equation:

$$\begin{aligned} &\forall\, x \geq 0:\; F_{hyd} = p_{in}\frac{1}{4}\mathrm{d}^2_{\mathrm{pist}}\pi - \mathrm{p}_{\mathrm{out}}\frac{1}{4}\mathrm{d}^2_{\mathrm{seat}}\pi\,, \\ &\forall\, x < 0:\; F_{hyd} = p_{in}\frac{1}{4}d^2_{pist}\pi - p_{out}\frac{1}{4}d^2_{seat}\pi - \left(p_{out} - \Delta p_{seat}\right)\frac{1}{4}\left(d^2_{pist} - d^2_{seat}\right)\pi \end{aligned} \tag{73}$$

where p_{in} and p_{out} are the input and output pressures and its seat, d_{pist} and d_{seat} are the diameters of piston and its seat and Δp_{seat} is the pressure drop at piston seat due to throttling.

Pressure drop at piston seat is calculated from the Bernoulli equation:

$$\Delta p_{seat} = \frac{1}{2}\rho\left(\frac{\dot{Q}}{A_{flow}}\right)^2, \tag{74}$$

where ρ is the fluid density, $\dot{Q}$ is the flow rate through piston seat and A_{flow} is the narrowest open flow area of piston seat. $\dot{Q}$ and A_{flow} are given by:

$$\dot{Q} = -\frac{\pi}{4}\left(d_{pist}^2 - d_{seat}^2\right)\dot{x}\,, \tag{75}$$

$$A_{flow} = -\pi d_{seat} x\,. \tag{76}$$

Damping force from squeezing fluid F_{damp} is defined as follows:

$$\begin{aligned} &\forall\, \dot{x} < 0 : F_{damp} = c_{fluid}\dot{x}\,, \\ &\forall\, \dot{x} \geq 0 : F_{damp} = 0, \end{aligned} \tag{77}$$

where c_{fluid} is the damping coefficient of squeezing fluid at piston seat. Clearly, F_{damp} is always zero at piston opening (forward motion out of seat) and, if $c_{fluid} \neq 0$, nonzero at piston closing (backward motion to seat).

Forces from input and output stops F_{in_st} and F_{out_st} are calculated in the same way as for the **Standard Piston** (refer to Equations 64, 67 and 65, 68).

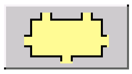
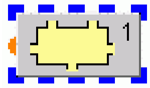
Viscous friction (shear) force F_{shear} (if any) is also calculated analogously to **the Standard Piston** (refer to Equation **236**).

Damping and stiffness coefficients of piston seat and stop and damping coefficient of squeezing fluid are specified directly in the input dialog of **SID Piston.**

Note that equation of motion of **SID Piston** (Equation 72) is similar to the equation of motion of **Standard Piston** (Equation 62) except for the calculation of hydraulic force and additional damping force from squeezing fluid. However, Equation 72 is in any case nonlinear because hydraulic force F_{hyd} depends on the pressure drop at piston seat Δp_{seat} and thus is a function of the piston velocity and coordinate ratio $\dot{x}/x$ as seen from Equations 74 to 76.

8. VOLUME

8.1. Standard Volume

Element Name:	**Standard Volume**	
Element Icon:		
Definition:	This element serves to define the properties of a standard Volume with rigid walls.	
Connecting pins:	standard pins: 10 (all hydraulic) special pins: 0 wire pins: 1	

☞ **Note:** **Standard Volume** element is not restricted to any geometric shape. If the **Volume** is attached to piston-type element(s), its volume is a variable and BOOST Hydsim will calculate its variation. If the initial volume is squeezed out by the piston motion, **BOOST Hydsim** will stop calculation and write out an error message.

Schematic of **Standard Volume** element is shown in Figure 61.

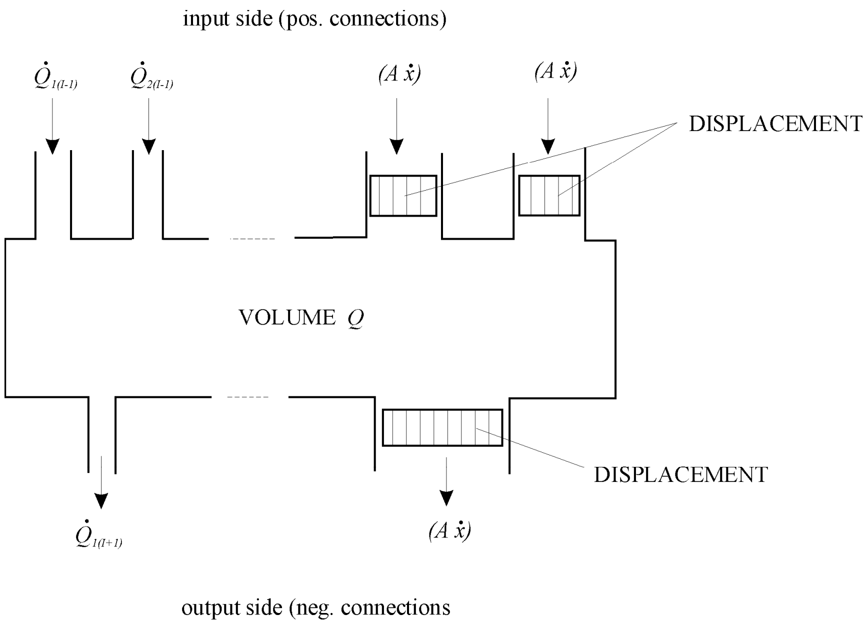


Figure 61: Schematic of Standard Volume

8.1.1. Input Parameters

Description of input data for the **Standard Volume**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.5).
Initial volume	Specify volume at start of calculation. If no piston-type element is attached to **Volume,** volume (initial volume) will stay same throughout the whole calculation. Only pressure will vary. If a piston-type element is attached to **Volume,** initial volume will change according to the position of attached piston(s). Volume and pressure will vary.
CAVITATION	
Vapor pressure	Specifies the fluid vapor pressure inside **Volume.** Normally, for vapor pressure a positive value must be specified. If it is set to 0 or not specified, the program assumes it to be undefined. In this case the cavity effects are neglected and negative pressures are possible. If the actual pressure drops down to the limit (vapor pressure), it is kept constant at the limiting value. To satisfy the mass conservation law, a vapor space (cavity) is then considered which has to be refilled before the pressure may increase again (refer to Section for more information).
Initial cavity	Specify cavity size at start of calculation.
Heat transfer to ambience	For this option, the rate of heat flow over the outer surface area of **Volume** (from the fluid inside **Volume** to the ambient environment) is calculated from the following equation: $\dot{Q} = h_{total} A_{out} \left(T_{fluid} - T_{ambient} \right)$, (78) where: *htotal* ……… overall heat transfer coefficient to outside *Aout* ………. outer surface area *Tfluid* …..… temperature of fluid in Volume *Tambient* … ambient temperature
Heat transfer coef.	Specify overall heat transfer coefficient to outside (htotal) between the fluid inside **Volume** and ambience.
Outer surface area	Specify outer surface area of **Volume** (Aout).
Ambient temperature	Specify ambient temperature around **Volume** (Tambient).

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 8.1.8).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)

8.1.2. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial condition:

Pressure:	Specify initial pressure in **Volume** at the beginning of calculation. All elements connected to **Volume** will use this initial pressure at start.
Temperature:	Specify initial temperature in **Volume** at the beginning of calculation. All elements connected to **Volume** will use this initial temperature.

☞ **Note:** To start calculation from equilibrium, all hydraulically connected **Volumes** must have the same initial pressure and temperature. If initial pressure and/or temperature is not specified under initials, they are set to 1 bar and 293.15 K.

8.1.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

vapor pressure:	SI Unit:	Pa (N/m^2)
heat transfer coef.:	SI Unit:	W/(m^{2}K)
ambient temperature:	SI Unit:	K

8.1.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the option box in front of output parameter must be activated.

Description of output parameters:

pressure	Pressure is calculated from Equation 79
actual volume	Actual volume is calculated from Equation 80
flow rate at volume input end /time	Sum of all flow rates at input end in **Time** domain.
flow rate at volume input end /angle	Sum of all flow rates at input end in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter **26.2.6.3**). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
flow rate at volume output end /time	Sum of all flow rates at output end in **Time** domain.
flow rate at volume output end /angle	Sum of all flow rates at output end in **Reference angle** domain or **Crank angle** domain.
vapor space from cavity	Vapor cavity calculated from equation (83).
max. pressure till selected point	Maximum volume pressure from beginning of calculation will be saved. It may be useful for e.g. Search Adjust Parameter (1D optimization).
fluid bulk modulus	Bulk modulus, either calculated internally or specified by the user.
fluid temperature	Fluid temperature either calculated internally or specified by the user according to the selected **Calculation task** (isothermal or thermal) in **Simulation | Mode** menu.
max. temperature till selected point	Maximum fluid temperature from beginning of calculation will be saved. It may be useful for e.g. Search Adjust Parameter (1D optimization).
thermal expansion coefficient	Fluid thermal expansion coefficient either calculated internally or specified by the user.
fluid density	Fluid density either calculated internally or specified by the user.
heat flow to ambience	Rate of heat flow over the outer area from the fluid to ambience (refer to equation 78).

8.1.5. Local Fluid Properties

Volume element (as any hydraulic element in BOOST Hydsim) may possess its own local fluid properties. If defined, these properties will replace the global fluid properties. Open the **Dialog Box** of local fluid properties by clicking the **Fluid Properties** button in the element input dialog box.

There are four options within the **Local Fluid Properties** dialog:

1. **Global (constant or variable) fluid properties** [default].
   Fluid properties are taken from the table of **Global Fluid Properties** (refer to Chapter 22.2.6.2 for more information).

2. **Local fluid properties from Property Database**
   Fluid properties are chosen from the menu in Property Database (refer to Chapter 22.2.6.5 for more information)
   - Fluid [name of specie]
   - Temperature
3. **Local (constant) fluid properties.**
   Five parameters of the fluid in the volume have to be specified:
   - Bulk modulus E [Pressure Units]
   - Reference density ρ_{ref} [Density Units]
   - Kinematic viscosity [Kinematic Viscosity Units]
   - Surface tension [Linear Stiffness Units]
   - Reference pressure p_{ref} [Stress Units]

> **Note:** Using this option, only isothermal calculation can be performed. Density is considered as pressure-dependent property $\rho = f(p)$ and is calculated by:
>
> $$\rho(p) = \rho_{ref} \cdot e^{\frac{1}{E}(p - p_{ref})}$$

4. **Local (variable) fluid properties.**
   The tabular values of fluid properties as a function of pressure have to be specified for the following properties:
   - Bulk modulus [Stress Units] :
     - there are two options for the bulk modulus specification:
       1. user-defined bulk modulus : $E = f(p)$
       2. internally calculated bulk modulus:

With the 1st option, bulk modulus is specified through the table as pressure-dependent property. This option is used if the button **Calculate bulk modulus internally from density-pressure function** is switched off (default). If this option is switched on, the bulk modulus column is grayed out and the bulk modulus value at each row is calculated as a product of the respective density and the partial derivative of density with respect to pressure at constant temperature:

$$E = \rho \frac{1}{\left(\frac{\partial \rho}{\partial p}\right)_T}$$

The above equation has to be satisfied in any case. For the 1st option, it is fully the user responsibility. For the 2nd option, partial derivative is calculated numerically. Hence accurate and smooth density-pressure function has to be specified in order to prevent numerical distortion.

- Density [Density Units] $\rho = f(p)$

- Kinematic viscosity [Kinematic Viscosity Units] $\nu = f(p)$
- Surface tension [Linear Stiffness Units] $\sigma = f(p)$

Intermediate values of variable fluid properties are linearly interpolated. Data extrapolation is not performed, as in case of global fluid properties. If pressure in the volume drops below the lowest pressure value given in the first row of the table, the values of bulk modulus, density, viscosity and surface tension from the first table row will be used.

Analogously, if pressure in the volume exceeds the highest pressure value given in the last row of the table, the values of bulk modulus, density, viscosity and surface tension from the last row will be used in calculation. The pressure values in the table of variable fluid properties must be arranged in ascending order.

☞ **Note:** Using this option, only isothermal calculation can be performed. All properties are considered as functions of pressure (but not temperature).

8.1.6. Governing Equations for Isothermal Flow

Pressure in **Standard Volume** at isothermal flow is calculated from the continuity equation:

$$\dot{p}(t) = \frac{E}{V(x)} \left\{ \sum_{i=1}^{n_1} \dot{Q}_i + \sum_{i=1}^{n_2} A_i \dot{x}_i - \sum_{j=1}^{l_1} \dot{Q}_j - \sum_{j=1}^{l_2} A_j \dot{x}_j \right\}, \quad (79)$$

$\dot{p}(t)$	pressure time derivative
E	bulk modulus of the fluid
$V(x)$	actual volume
$\dot{Q}_i$ and $\dot{Q}_j$	flow rates through i-th input and j-th output connections to non-piston-type elements
n_1 and l_1	number of hydraulic connections to non-piston-type elements on input and output ends
n_2 and l_2	number of hydraulic connections to piston-type elements on input and output ends
A_i and A_j	cross-sectional areas of piston-type elements connected to Volume by i-th input and j-th output connection, respectively
$\dot{x}_i$	velocity of piston-type element connected to Volume by i-th input connection
$\dot{x}_j$	the velocity of piston-type element connected to Volume by j-th output connection

Standard Volume may have only hydraulic connections. Depending on the connected elements, these connections are divided into piston-type or non-piston-type. Piston-type connections imply the connection to the following elements: **Piston, SID Piston** (Piston group), **Plunger, RPD Pump** (Pump group), **Solenoid** Bodies (Solenoid group) and any **Needle** element (Needle group). All other hydraulic connections to **Volume** are non-piston-type (to the elements of **Boundary, Line, Bend, Junction, Port, Valve, Throttle, Orifice** and **Nozzle** groups).

Actual volume is calculated from the following equation:

$$V(x) = V_0 - \sum_{i=1}^{n_2} A_i x_i + \sum_{j=1}^{l_2} A_j x_j \quad , \tag{80}$$

where *V0* is the initial volume and x_i and x_j are the displacements of piston-type elements connected to **Volume** by *i-th* input and *j-th* output connection, respectively.

Equation 80 shows that positive displacement of the connected piston-type element on input end will compress the volume, while positive piston displacement on output end will expand the volume. Obviously, if there are no hydraulic connections to piston-type elements neither on input nor on output end, the volume will remain constant.

8.1.6.1. Cavitation model

If at a certain time instant pressure *p(t)* in the **Volume** drops down to vapor pressure *pvap,* then it is kept constant at the vapor pressure. The continuity equation *(*79*)* implies that in this case the pressure derivative $\dot{p}(t) = 0$, that is:

$$\forall\ p(t) \le p_{vap} :\ \frac{E}{V(x)}\left\{\sum_{i=1}^{n_1} \dot{Q}_i + \sum_{i=1}^{n_2} A_i \dot{x}_i - \sum_{j=1}^{l_1} \dot{Q}_j - \sum_{j=1}^{l_2} A_j \dot{x}_j\right\} = 0 . \tag{81}$$

Mass conservation equation (*81)* cannot be satisfied in this form because the term in brackets is nonzero. To formally satisfy it, a vapor cavity volume $V_c(t)$ is introduced:

$$\sum_{i=1}^{n_1} \dot{Q}_i + \sum_{i=1}^{n_2} A_i \dot{x}_i - \sum_{j=1}^{l_1} \dot{Q}_j - \sum_{j=1}^{l_2} A_j \dot{x}_j + \dot{V}_c(t) = 0 , \tag{82}$$

where $\dot{V}_c(t)$ is the time derivative of cavity volume (evaporation rate).

If, e.g., at time *t1* pressure in **Volume** *p(t)* becomes less or equal vapor pressure *pvap,* the cavity volume at time *t>t1* is calculated from:

$$V_c(t) = \int_{t_1}^{t}\left\{\sum_{j=1}^{l_1} \dot{Q}_j + \sum_{j=1}^{l_2} A_j \dot{x}_j - \sum_{i=1}^{n_1} \dot{Q}_i - \sum_{i=1}^{n_2} A_i \dot{x}_i\right\} dt. \tag{83}$$

Pressure *p(t)* may increase only when cavity volume $V_c(t)$ becomes zero again.

☞ **Note:** Equation 82 is not mathematically rigorous. Hence, the cavitation model described above is just a simplified engineering approximation of complex two-phase flow phenomenon. In many cases it yields satisfactory qualitative results. However, exact treatment of cavitation with this model is not feasible. Moreover, it may affect the stability of numerical integration if vapour cavity tends to become large.

8.1.7. Governing Equations for Thermal Flow

Pressure and temperature in **Standard Volume** at thermal flow is calculated from the continuity equation, equation of state for compressible fluid and the conservation of energy.

Continuity equation (conservation of mass) is given by:

$$\sum \dot{m} = \sum \left(\frac{dm}{dt} \right)_i = \sum \left(\rho \dot{q} \right)_i \ , \tag{84}$$

where:

$\dot{m}$ .... rate of mass change in Volume

ρ .... fluid density

$\dot{q}$ .... volumetric flow rate

i .... index for inflows and outflows (inflows have positive sign and outflows – negative sign)

Equation of state for the compressible fluid has the form:

$$\frac{d\rho}{\rho} = \frac{1}{E} dp - \alpha dT \ . \tag{85}$$

For pressure-invariant bulk modulus and temperature-invariant thermal expansion coefficient, equation (85) can be solved analytically for density:

$$\rho = \rho_0 e^{\frac{1}{E}(p-p_0)} \rho_0 e^{-\alpha(T-T_0)} = \rho_0 e^{\frac{1}{E}(p-p_0)-\alpha(T-T_0)} \ . \tag{86}$$

Equation of energy conservation follows from the first law of thermodynamics:

$$\frac{dU}{dt} = \frac{dH_{in}}{dt} - \frac{dH_{out}}{dt} + \frac{dW}{dt} - \frac{dQ}{dt} \ , \tag{87}$$

where:

$\frac{dU}{dt}$ .... rate of change of internal energy of fluid

$\frac{dH_{in}}{dt}, \frac{dH_{out}}{dt}$ .... enthalpy of inflows and outflows, respectively

$\frac{dW}{dt}$ .... rate of work done on the fluid in Volume

$\frac{dQ}{dt}$ .... rate of heat flow over the outer surface area

Equation (87) can be expressed in the simplified form:

$$\frac{d}{dt}(mcT) = \sum \dot{m}_i c_i T_i + p\left(\sum \dot{q}_i - \dot{V}\right) - \dot{Q}, \tag{88}$$

Substituting energy equation *(88)* and continuity equation (84*)* into equation of state (85*),* we obtain:

$$\dot{p}(t) = \frac{E}{m}\left(\dot{m} - \rho\dot{V}\right) + E\alpha\dot{T},$$
$$\dot{T}(t) = \frac{1}{mc}\left[\sum (c_i T_i - cT)\dot{m}_i + p\left(\sum \dot{q}_i - \dot{V}\right) - \dot{Q}\right] \tag{89}$$

$\dot{p}(t)$	pressure time derivative
$\dot{T}(t)$	temperature time derivative
E	fluid bulk modulus
ρ_0	fluid density at reference pressure p_0 and temperature T_0
m	fluid mass
$\dot{m}$	rate of fluid mass change in volume ($= \sum \dot{m}_i$)
α	thermal expansion coefficient of fluid
c	specific heat capacity of fluid
c_i	specific heat capacity of *i-th* inflow / outflow
T_i	temperature of *i-th* inflow / outflow
T	fluid temperature
T_0	reference temperature
p_0	reference pressure
p	fluid pressure
$\dot{q}_i$	volumetric flow rate of *i-th* inflow / outflow
$\dot{V}$	rate of Volume change due to compression/expansion
$\dot{Q}$	rate of heat flow from fluid to ambience (refer to equation 78)

Rate of volume change due to compression/expansion is calculated from the following equation:

$$\dot{V} = \sum_{j=1}^{l_2} A_j \dot{x}_j - \sum_{i=1}^{n_2} A_i \dot{x}_i, \tag{90}$$

where $\dot{x}_i$ and $\dot{x}_j$ are the velocities of piston-type elements connected to **Volume** by *i-th* input and *j-th* output connection, respectively.

All equations in this chapter are valid for the liquid phase only. In case of cavitation (pressure drop to vapor pressure) the fluid temperature is simply kept constant together with the vapor pressure.

8.1.8. Generation of Excitation Force

For the calculation of forces from hydraulic elements such as volumes, which are transferred to the housing, two different cases have to be distinguished.

For a volume that is not in contact with the piston type elements it is assumed that the pressure is equally distributed inside the volume, and thus the resultant integral force on the volume walls is zero.

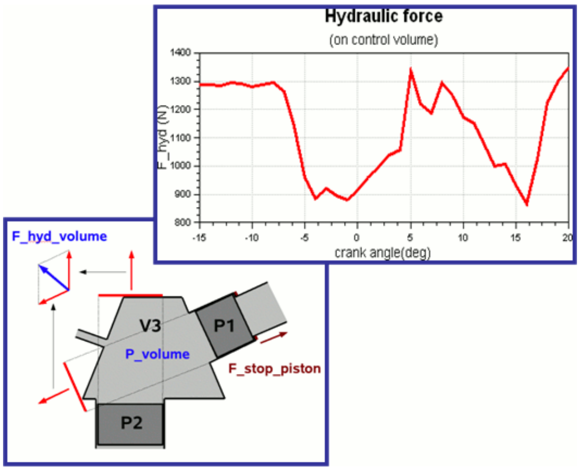
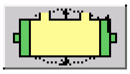
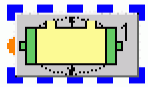


Figure 62: Volume Excitation

A volume that is in direct contact with one or more piston type elements (see Figure 62), pressure within the volume acts not only on the volume walls, but also on the pressurized side of the pistons. Since the pressure force at the piston passes onto the rest of the multi-body system through the piston springs and its contacts, the unbalanced force remains on the other side of the volume. It acts on volume walls and has the same amount but the opposite direction to pressure force on the piston. At such positions hydraulic force may have significant influence on the generation of excitation force.

8.2. Compliant Volume

Element Name:	**Compliant Volume**	
Element Icon:		
Definition:	This dialog serves to define a Volume with compliant walls (cylindrical or spherical). The sketch of Volume of cylindrical shape with possible connections is shown in Figure 62. Volume of spherical shape is depicted in Figure 63.	
Connecting pins:	standard pins: 10 (all hydraulic) special pins: 0 wire pins: 1 Among these, up to two connections to piston-type elements are possible if **Compliant Volume** has a cylindrical shape (refer to Figure 62).	

> **Note:** For **Compliant Volume,** the effects of volume wall distension are encountered at each time step according to internal pressure. This volume must have a form of either a cylinder or a sphere. Up to two piston-type elements can be connected to the ends of a volume of cylindrical shape.

The schematic of **Compliant Volume** is shown in Figure 62.

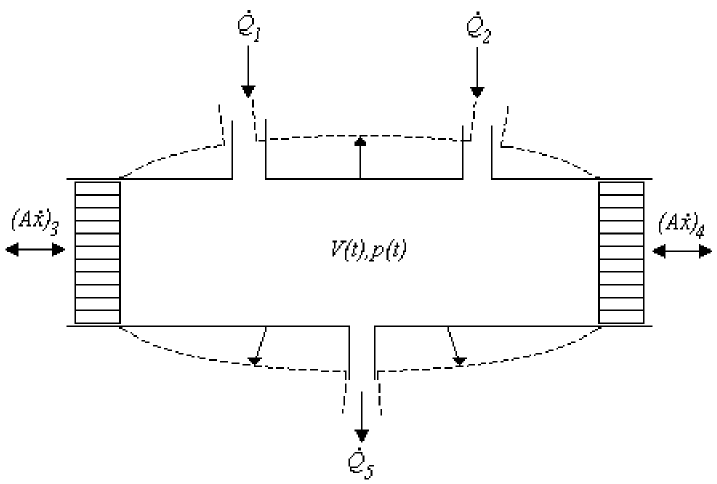


Figure 63: Compliant Volume - Cylindrical Shape with Two Pistons

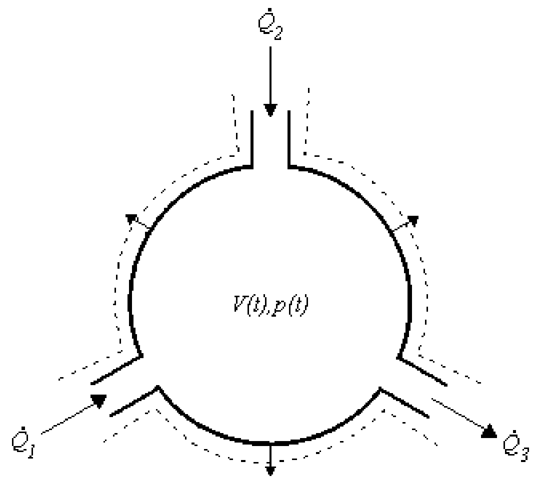


Figure 64: Compliant Volume - Spherical Shape

8.2.1. Input Parameters

Description of input data for the **Compliant Volume**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.5).
Solid Properties	Press this bar to specify the local solid properties of Volume walls (refer to Section 8.2.2).
Inner diameter:	Specify inner volume diameter (of a cylinder or a sphere).
Wall thickness:	Specify the thickness of volume walls.
Cylindrical shape:	This button defines that the shape of **Volume** is cylindrical.
Cylinder length:	Specifies the length of cylinder. If piston-type elements are connected to the **Volume** end(s), initial length between pistons' inner surface or piston inner surface and volume end has to be specified. This input is active only if the **Cylindrical shape** button is checked.
Spherical shape:	This button defines that the shape of **Volume** is spherical.
Cavitation:	
Vapor pressure:	Specifies the fluid vapor pressure inside volume. Normally, for vapor pressure a positive value must be specified. If it is set to 0 or not specified, the program assumes it to be undefined. In this case the cavity effects are neglected and negative pressures are possible.

	If the actual pressure drops down to the limit (vapor pressure), it is kept constant at the limiting value. To satisfy the mass conservation law, a vapor space (cavity) is then considered which has to be refilled before the pressure may increase again (refer to Section 8.1.6.1 for more information concerning the cavitation model).
Initial cavity:	Specify cavity size at start of calculation.
Wall Deformation:	This button activates the calculation of walls deformation. Input of `Ambient pressure` is active only if the toggle button **Wall Deformation** is checked. If the toggle button **Wall Deformation** is deactivated, **Compliant Volume** becomes a **Standard Volume** of a specific shape.
Ambient pressure:	Specify pressure outside of compliant volume wall *pout.*
Heat Transfer through Walls:	For this option, the rate of heat flow over outer surface area of the Volume (from the fluid to ambient environment) will be calculated.
Ambient temperature	Specify ambient temperature around the **Volume** (Tambient). **Note:** Air is assumed as an ambient environment.
Program-calculated heat convection coef.	If this button is active, the internal calculation of both heat transfer coefficients (α1 and α2) will be used. **Note:** Program – calculated heat convection coef. option is available only for Cylindrical shape.
User-defined heat convection coef.	Click to use the user-defined heat convection coefficients from fluid inside **Volume** to wall and from wall to ambience (instead of using the program-calculated overall heat transfer coefficient *htotal).*
Fluid to solid wall	Specify heat transfer coefficient (α1) between fluid inside **Volume** and Volume walls.
Wall to ambience	Specify heat transfer coefficient (α2) between Volume walls and ambience.

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 8.1.8).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)

8.2.2. Local Solid Properties

This dialog serves to define the local solid properties.
Local solid properties can be **constant** or **variable** (temperature-dependent). Variable solid properties can be selected only through **Property Database**.

Description of input data:

GLOBAL SOLID PROPERTIES	Solid properties are taken from the table of **Global Solid Properties** dialog box (default).
SOLID PROPERTIES FROM PROPERTY DATABASE	Click to activate the selection of solid properties from the Property Database (refer to Chapter 25.2.4.8 for more information).
Solid name	Specify solid from the menu list.
Temperature	Specify constant solid temperature.
CONSTANT SOLID PROPERTIES	Click to activate constant solid properties.
Density	Specify the solid density as local solid property. It is kept constant throughout the whole calculation.
Young's modulus	Specify the Young's modulus of solid as local solid property. It is kept constant throughout the whole calculation.
Poisson's ratio	Specify the Poisson's ratio of solid as local solid property. It is kept constant throughout the whole calculation.
Thermal conductivity	Specify the thermal conductivity of solid as local solid property. It is kept constant throughout the whole calculation.
Specific heat capacity	Specify the specific heat of solid as local solid property. It is kept constant throughout the whole calculation.

8.2.3. Initial Conditions

The **Dialog Box of Initial Condition** (pressure) for **Compliant Volume** is identical to that of **Standard Volume** (refer to Section 8.1.2).

8.2.4. Modifiable Parameters

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is checked, Modification Table must be defined (refer to Section 25.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

vapor pressure:	SI Unit:	Pa
inner diameter	SI Unit:	m
wall thickness:	SI Unit:	m
ambient pressure:	SI Unit:	Pa
ambient temperature:	SI Unit:	deg K
heat convection fluid-wall:	SI Unit:	$W/m^2.K$
heat convection wall-ambience:	SI Unit:	$W/m^2.K$

8.2.5. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter must be checked.

Description of output parameters:

fluid pressure	Pressure is calculated from Equation 91 (refer to Section 8.2.6 for more information).
actual volume	Actual volume is calculated from Equation 92 (refer to Section 8.2.6 for more information).
actual diameter	Depending on the **Volume** shape, formulas for calculation of actual diameter are given in Section 8.2.6.
flow rate at volume input end /time	Sum of all flow rates at input end in **Time** domain.
flow rate at volume input end /angle	Sum of all flow rates at input end in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter 25.2.7.4). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
flow rate at volume output end /time	Sum of all flow rates at output end in **Time** domain.
flow rate at volume output end /angle	Sum of all flow rates at output end in **Reference angle** domain or **Crank angle** domain.
vapor space from cavity	Vapor cavity calculated from equation (83).

max. pressure till selected point	Maximum volume pressure from beginning of calculation will be saved. It may be useful for e.g. Search Adjust Parameter (1D optimization).
fluid bulk modulus	Bulk modulus, either calculated internally or specified by the user.
fluid temperature	Fluid temperature either calculated internally or specified by the user according to the selected **Calculation task** (isothermal or thermal) in **Simulation | Mode** menu.
max. temperature till selected point	Maximum fluid temperature from beginning of calculation will be saved. It may be useful for e.g. Search Adjust Parameter (1D optimization).
thermal expansion coefficient	Fluid thermal expansion coefficient either calculated internally or specified by the user.
fluid density	Fluid density either calculated internally or specified by the user.
heat flow to ambience	Rate of heat flow over the outer area from the fluid to ambience.
heat transfer coefficient	Overall heat transfer coefficient from the fluid to ambience (refer to equations 100 and 104).
inner wall temperature	Inner wall temperature, either calculated internally or set to the fluid temperature, depending the selected **Calculation task** (isothermal or thermal) in **Simulation | Mode** menu and defined Volume shape.
outer wall temperature	Outer wall temperature, either calculated internally or set to the fluid temperature, depending the selected **Calculation task** (isothermal or thermal) in **Simulation | Mode** menu and defined Volume shape.

8.2.6. Governing Equations for Isothermal Flow

Pressure in Compliant Volume at isothermal flow is calculated from the continuity equation:

$$\dot{p}(t) = \frac{E}{V(x,p)} \left\{ \sum_{i=1}^{n_1} \dot{Q}_i + \sum_{i=1}^{n_2} A_i \dot{x}_i - \sum_{j=1}^{l_1} \dot{Q}_j - \sum_{j=1}^{l_2} A_j \dot{x}_j \right\},\ n_2 + l_2 \le 2\,, \tag{91}$$

where:

$\dot{p}(t)$	pressure time derivative
E	bulk modulus of the fluid
$V(x,p)$	actual volume
$\dot{Q}_i$ and $\dot{Q}_j$	flow rates through *i-th* input and *j-th* output connections to non-piston-type elements
n_1 and l_1	number of hydraulic connections to non-piston-type elements on input and output ends
n_2 and l_2	number of hydraulic connections to piston-type elements on input and output ends

A_i and A_j	cross-sectional areas of piston-type elements connected to Volume by *i-th* input and *j-th* output connection, respectively
$\dot{x}_i$	velocity of piston-type element connected to Volume by *i-th* input connection
$\dot{x}_j$	velocity of piston-type element connected to Volume by *j-th* output connection

Piston-type elements can be connected only to Compliant Volume of cylindrical shape, and only along the cylinder axis as shown in Figure 62.

Compliant Volume may have only hydraulic connections. Depending on the connected elements, these connections are divided into piston-type or non-piston-type. Piston-type connection imply the connection to the following elements: Piston, SID Piston (Piston group), Plunger, RPD Pump (Pump group), Solenoid bodies (Solenoid group), Valve bodies (Valve group) and any Needle element (Needle group). All other hydraulic connections to Volume are non-piston-type (to the elements of Boundary, Line, Port, Throttle, Orifice and Nozzle groups). Note that, contrary to Standard Volume, not more than two piston-type elements can be connected to Compliant Volume ($n_2 + l_2 \le 2$).

Actual volume is calculated from the following equation:

$$V(x,p) = V_0 - \sum_{i=1}^{n_2} A_i x_i + \sum_{j=1}^{l_2} A_j x_j + \Delta V(p), \tag{92}$$

where V_0 is the initial volume, x_i and x_j are the displacements of piston-type elements connected to Compliant Volume by *i-th* input and *j-th* output connection, respectively, and $\Delta V(p)$ is the volume distension due to the pressure variation.

Equation 92 shows that positive displacement of piston-type element connected to the input end of Compliant Volume will compress the volume, while positive displacement of piston-type element connected to the output end of Volume will expand the volume.

Calculation of the volume distension depends on its shape. For a cylindrical volume, the distension without consideration of cylinder elongation is given by:

$$\Delta V(p) = \frac{\pi}{4} \Delta D_{in} \left(\Delta D_{in} + 2D_{in}\right) \left(L - \sum_{i=1}^{n_2} x_i + \sum_{j=1}^{l_2} x_j \right), \tag{93}$$

where D_{in} is the initial inner diameter (at equal inner and outer pressures), ΔD_{in} is the distension of the inner diameter and L is the initial length of the cylinder (at zero positions of the connected piston-type elements if any).

For a spherical volume, the distension is calculated from the formula:

$$\Delta V(p) = \frac{\pi}{6} \left(\left(D_{in} + \Delta D_{in}\right)^3 - D_{in}^3 \right). \tag{94}$$

To calculate the distension (increase) of inner diameter ΔD_{in}, we have to distinguish between the following two cases2:

- Thick-walled volumes (thick shells of revolution)

$$\frac{D_{out}}{t} \leq 10,$$

where D_{out} is the initial outer diameter of the volume and t is the wall thickness.

Cylindrical shape:

$$\Delta D_{in\_in} = \frac{p_{in} D_{in}}{E_w} \left(\frac{D_{out}^2 + D_{in}^2}{D_{out}^2 - D_{in}^2} + \nu \right),$$

$$\Delta D_{in\_ext} = \frac{-p_{ext}}{E_w} \frac{2 D_{in} D_{out}^2}{D_{out}^2 - D_{in}^2},$$

$$\Delta D_{in} = \Delta D_{in\_in} - \Delta D_{in\_ext} \tag{95}$$

where E_w and ν are the Young's modulus and Poisson's ratio of the wall material.

Spherical shape:

$$\Delta D_{in\_in} = \frac{p_{in} D_{in}}{E_w} \left(\frac{(1-\nu)(D_{out}^3 + 2D_{in}^3)}{2(D_{out}^3 - D_{in}^3)} + \nu \right),$$

$$\Delta D_{in\_ext} = \frac{-p_{ext} D_{in}}{E_w} \frac{3(1-\nu) D_{out}^3}{2(D_{out}^3 - D_{in}^3)},$$

$$\Delta D_{in} = \Delta D_{in\_in} - \Delta D_{in\_ext} \tag{96}$$

[2] Raymond R. Roark, Warren C. Young, Formulas for Stress and Strain, 5th Edition, McGraw-Hill.

- Thin-walled volumes (pressure vessels)

$$\frac{D_{out}}{t} > 10 .$$

Cylindrical shape:

$$\Delta D_{in} = \frac{(p_{in} - p_{ext}) D_{in}^2}{4 E_w t} \tag{97}$$

Spherical shape:

$$\Delta D_{in} = \frac{(p_{in} - p_{ext}) D_{in}^2}{8 E_w t} (1 - \nu) \tag{98}$$

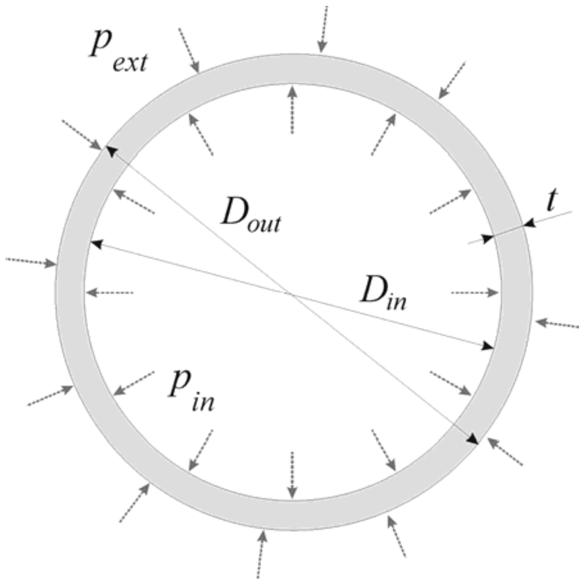


Figure 65: Cylindrical Compliant Volume with Uniform Pressure

8.2.7. Governing Equations for Thermal Flow

Pressure and temperature in **Compliant Volume** for thermal flow is calculated from the same equations as in **Standard Volume** (refer to Section 8.1.7. Note that for **Compliant Volume,** the volume change due to wall deformation *(ΔD_{in})* is usually quite small. Hence it is not included into the calculation of the rate of volume change due to compression/expansion *(d_V/d_t)*.

8.2.7.1. Heat Transfer Calculation

The heat transfer coefficients from fluid to wall *(α_1)* and from wall to ambience *(α_2)* can be either program-calculated or user-defined. Program-calculated coefficient option is available for the cylindrical volume shape, only.

Calculation of the heat transfer coefficients at inner and outer surface area and heat transfer from the fluid to the ambience depend on the Volume shape:

- Cylindrical shape:

  If **Program – calculated heat convection coef.** option is active, individual heat transfer coefficients (inner - α_1 and outer - α_2) are calculated according to the following equations:

$$\alpha_1 = \frac{\lambda_1 Nu_1}{D_{in}},$$

$$\alpha_2 = \frac{\lambda_2 Nu_2}{D_{out}},$$

where:

λ_1 , λ_2	thermal conductivity of the fluid and ambience (air), respectively
D_{in}, D_{out}	inner and outer diameters of the Volume, respectively
Nu_1, Nu_2	Nusselt number at inner and outer walls, respectively

$$Nu_1 = 1$$

$$Nu_2 = \left\{ 0.60 + 0.387 \cdot Ra_D^{1/6} \cdot \left[1 + \left(\frac{0.559}{\Pr} \right)^{\frac{9}{16}} \right]^{-\frac{8}{27}} \right\}^2 \tag{99}$$

The above equation for Nusselt number at outer wall is valid for:

0 < Pr <∞

10-5 < RaD < 1012,

where Ra_D is Rayleigh number given by:

$$Ra_D = Gr \cdot \Pr.$$

Here *Pr* is Prandtl number:

$$\Pr = \frac{c_{p(2)} \cdot \mu_2}{\lambda_2},$$

where:

$c_{p(2)}$	specific heat at constant pressure of the ambience (air)
μ_2	dynamic viscosity of the ambience
λ_2	thermal conductivity of the ambience

Gr – Grashof number:

$$Gr = \frac{D_{out}^3 \rho_2^2 \beta_2 \left(T_2 - T_{ambience}\right)}{\mu^2} g ,$$

where:

D_2 …………………. outer diameter

ρ_2 …………………… density of the ambience (air)

β_2 …………………… thermal expansion coefficient of the ambience

T2 ………………….. outer wall temperature

Based on the individual heat transfer coefficients and thermal conductivity of the wall, the overall heat transfer coefficient at the outer surface from the fluid to ambience is calculated according to the following equation:

$$\frac{1}{h_{total}} = \frac{D_{out}}{\alpha_1 D_{in}} + \frac{D_{out}}{2\lambda \log\left(\frac{D_{in}}{D_{out}}\right)} + \frac{1}{\alpha_2} , \tag{100}$$

where:

h_{total} …………….. overall heat transfer coefficient to outside

α_1 ……………….. inner heat transfer coefficient (from fluid to walls)

α_2 ……………… outer heat transfer coefficient (from walls to ambience)

λ ………….. wall thermal conductivity

There are two different calculation methods for the wall temperature. Their choice depends on the Biot number:

$$Bi = \frac{\alpha_1 R_{in}}{2\lambda} ,$$

where:

R_{in} ……………… inner Volume radius

If Biot number is less than 0.1, the so-called lumped-capacity solution is used: it is based on the transient heat conduction calculation. Otherwise, the steady-state calculation of the heat conduction through the Volume wall is used:

- Steady-state calculation (Bi > *0.1):*

  The rate of heat flow from the fluid in Volume to ambience is calculated by:

$$\dot{Q} = h_{total} \cdot A_2 \cdot \left(T_{fluid} - T_{ambient}\right) , \tag{101}$$

where:

A_2 .... outer Volume surface area.

Inner and outer wall temperatures are calculated by:

$$T_1 = T_{fluid} - \frac{\dot{Q}}{\alpha_1 A_1},$$

$$T_2 = T_{air} + \frac{\dot{Q}}{\alpha_2 A_2},$$

where:

T_1 .... inner wall temperature
T_2 .... outer wall temperature
A_1 .... inner Volume surface area.

- Lumped-capacity solution (B < 0.1):

  Constant temperature through the wall is assumed, i.e. inner wall temperature is equal to the outer wall temperature.

  Wall temperature is calculated from the following equation:

$$T = \frac{(B \cdot T_0 - A) \cdot e^{-B\Delta t}}{B} + \frac{A}{B}, \qquad (102)$$

$$A = \frac{T_{fluid}}{\tau_1} + \frac{T_{air}}{\tau_2}, \quad B = \frac{\tau_1 + \tau_2}{\tau_1 \tau_2},$$

$$\tau_1 = \frac{\rho c V}{\alpha_1 A_1}, \quad \tau_2 = \frac{\rho c V}{\alpha_2 A_2},$$

where:

τ_1, τ_2 .... time constants
ρ .... density of the wall material
c .... specific heat of the wall material
V .... volume of the wall material
T_0 .... wall temperature at previous time step
Δt .... calculation time step

The rate of heat flow from the fluid to ambience is calculated as the rate of heat flow from the fluid to the wall:

$$\dot{Q} = \alpha_1 \cdot A_1 \cdot (T_{fluid} - T_1). \qquad (103)$$

- Spherical shape:

  User- defined heat convection coef. option is available, only.

  The overall heat transfer coefficient is calculated as:

$$\frac{1}{h_{total}} = \frac{D_{out}^2}{\alpha_1 D_{in}^2} + \frac{(D_{out} - D_{in})D_{out}}{2\lambda D_{in}} + \frac{1}{\alpha_2}, \tag{104}$$

where:

htotal	overall heat transfer coefficient to outside
$\alpha 1$	user-defined inner heat transfer coefficient (from fluid to walls)
$\alpha 2$	user-defined outer heat transfer coefficient (from walls to ambience)
λ	wall thermal conductivity

The rate of heat flow from the fluid in Volume to ambience is calculated by:

$$\dot{Q} = h_{total} \cdot A_2 \cdot \left(T_{fluid} - T_{ambient}\right), \tag{105}$$

where:

A_2	outer Volume surface area.

Inner and outer wall temperatures are calculated as follows:

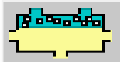
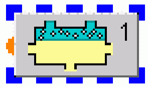
$$T_1 = T_{fluid} - \frac{\dot{Q}}{\alpha_1 A_1},$$

$$T_2 = T_{air} + \frac{\dot{Q}}{\alpha_2 A_2},$$

where:

T_1	inner wall temperature
T_2	outer wall temperature
A_1	inner Volume surface area.

8.3. Two-Phase Volume

Element Name:	**Two-phase Volume**	
Element Icon:		
Definition:	This element serves to define the properties of a two-phase Volume with bubble dynamics or fluid-gas mixture.	
Connecting pins:	standard pins: 10 (all hydraulic) special pins: 0 wire pins: 1	

Note: Currently the Variable-step solver does not support this element.

Note: **Volume** element may have an arbitrary shape. If the **Two-phase Volume** is attached to piston-type elements, its volume is a variable and BOOST Hydsim will calculate its variation. If initial volume is squeezed by piston motion, **BOOST Hydsim** will stop calculation and write out an error message.

Note: If **Two-phase Volume** is connected via **SAC Nozzle (Extended model)** element with **Gas Pressure/Temperature Boundary**, then gas inflow through the nozzle is accounted for in the fluid-gas mixture calculation.

8.3.1. Input Parameters

Description of input data for the **Two-phase Volume**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties. (refer to Section 8.1.5).
Initial volume	Specify volume at start of calculation. If no piston-type element is attached to **Volume,** volume (initial volume) will stay same throughout the whole calculation. Only pressure will vary. If a piston-type element is attached to **Volume,** initial volume will change itself according to positions of attached pistons. Volume and pressure will vary.
Gas bubble Dynamics	If this option is used, BOOST Hydsim will dynamically calculate void fraction according to the volume pressure (refer to Section 8.3.6.1).

☞ **Note:** **Gas Bubble Dynamics** is recommended for small gas fraction (cavitation).

Reference pressure in fluid	Specify reference pressure p_0 at which void fraction and bubble radius are defined.
Void fraction at reference pressure	Specify the gas fraction β_0 inside fuel + gas mixture at reference pressure p_0 .
Bubble radius at reference pressure	Specify the bubble radius R_0 at reference pressure.
fluid-gas mixture	If this option is used, BOOST Hydsim will calculate the parameters of the two-phase (liquid and gas) mixture, including mass and void fraction of gas.
Mass fraction of gas	Click to activate initial mass fraction.
Initial mass fraction	Specify initial mass fraction of gas inside fluid-gas mixture β_m.
Initial volumetric void fraction	Click to activate table of initial volumetric void fraction $\beta_v = f(p)$. BOOST Hydsim will take initial volumetric void fraction from this table according to the initial pressure in Volume.
Volumetric void fraction table	Specify pressure in ascending order in first column. Specify volumetric void fraction in second column. For pressures higher than the highest value specified, the highest pressure value from the table is used. For pressures lower than the lowest value specified, the lowest pressure value from the table is taken (kept constant as long as pressure is out of table range).
Gas Properties	Press this bar to specify the isentropic compression/expansion of gas. (refer to Section 8.3.5).
Heat transfer to ambience	For this option, the rate of heat flow over the outer surface area of **Two-phase Volume** (from the fluid inside volume to the ambient environment) is calculated from the following equation: $\dot{Q} = h_{total} A_{out} \left(T_{fluid} - T_{ambient}\right)$ (106) where: h_{total} ……… overall heat transfer coefficient to outside A_{out} ………. outer surface area T_{fluid} ………. temperature of fluid in Volume $T_{ambien}t$ ……. ambient temperature
Heat transfer coef.	Specify overall heat transfer coefficient to outside (htotal) between the fluid inside **Volume** and ambience.
Outer surface area	Specify outer surface area of **Volume** (Aout).
Ambient temperature	Specify ambient temperature around **Volume** (Tambient).

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 8.1.8).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)

8.3.2. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial condition:

Pressure:	Specify initial pressure in the **Volume** at the beginning of calculation. All elements attached to **Volume** will take this initial pressure.
Temperature:	Specify initial temperature in **Volume** at the beginning of calculation. All elements connected to **Volume** will use this initial temperature.

☞ **Note:** To start calculation from equilibrium, all hydraulically connected **Volumes** must have the same initial pressure and temperature. Default initial pressure and temperature are set to 1 bar and 20 deg C.

8.3.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

mass fraction of gas:	SI Unit:	[-]
heat transfer coef.:	SI Unit:	$W/(m^2K)$
ambient temperature:	SI Unit:	K

8.3.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option box in front of output parameter must be activated.

Description of output parameters:

pressure in fluid	Pressure is calculated from Equation 107 (refer to Section 8.3.5).
actual volume	Actual volume is calculated from Equation 108 (refer to Section 8.3.5).
flow rate at volume input end /time	Sum of all flow rates at input end in **Time** domain.
flow rate at volume input end /angle	Sum of all flow rates at input end in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter 26.2.6.3). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
flow rate at volume output end /time	Sum of all flow rates at output end in **Time** domain.
flow rate at volume output end /angle	Sum of all flow rates at input end in **Reference angle** domain or **Crank angle** domain.
max. pressure till selected point	Maximum volume pressure from beginning of calculation will be saved. It may be useful for e.g. Search Adjust Parameter (1D optimization).
gas bubble radius	Gas bubble radius is calculated from Equations 109 or Equation 110 (refer to Section 8.3.6.1).
void fraction	Volumetric void fraction is calculated either from Equations for active **Gas Bubble Dynamics** option (refer to Section 8.3.6.1) or from Equations for active **Fluid-gas Mixture** option (refer to Section8.3.6.2).
bulk modulus of 2-phase mixture	Bulk modulus of two-phase mixture is calculated from Equation 114 for active **Gas Bubble Dynamics** option (refer to Section 8.3.6.1) or from Equation 123 for active **Fluid-gas Mixture** option.
density of 2-phase mixture	Density of two-phase mixture is calculated from Equation 113 for active **Gas Bubble Dynamics** option (refer to Section 8.3.6.1) of from Equation 120 for active **Fluid-gas Mixture** option.
gas density (fluid-gas mixture only)	Gas density is calculated from Equation 117 only for **Fluid-gas Mixture** option (refer to Section 8.3.6.1).
Pressure inside bubble	Pressure inside bubble is calculated from Equation 115 for active **Gas Bubble Dynamics** option (refer to Section 8.3.6.1).
fluid temperature	Fluid temperature either calculated internally or specified by the user according to the selected **Calculation task** (isothermal or thermal) in **Simulation | Mode** menu.
max. temperature till selected point	Maximum fluid temperature from beginning of calculation will be saved. It may be useful for e.g. Search Adjust Parameter (1D optimization).

liquid thermal expansion coefficient	Thermal expansion coefficient of liquid phase either calculated internally or specified by the user.
liquid bulk modulus	Bulk modulus of liquid phase, either calculated internally or specified by the user.
liquid density	Liquid density either calculated internally or specified by the user.
heat flow to ambience	Rate of heat flow over the outer area from the fluid to ambience (refer to equation 106).
mass fraction of gas	Mass fraction of gas is calculated from Equation 116 for active **Fluid-gas Mixture** option.

8.3.5. Local Gas Properties (Two-Phase Volume)

If **Fluid-Gas Mixture** option is active, initial gas fraction (mass or volumetric) inside the fluid-gas mixture needs to be specified. It has to be defined for the isentropic compression/expansion of ideal gas.

The input of gas properties for **Fluid-Gas Mixture** option depends on the **Calculation task** (isothermal or thermal) in **Simulation | Mode** menu.

There are three options within the **Local Gas Properties** dialog:

1. **Global gas properties** [default].
   Gas properties are taken from the table of **Global Gas Properties** (refer to 24.2.5.5 for more information).
2. **Gas properties from Property Database**
   Gas properties are chosen from the menu in Property Database (refer to Chapter 24.2.5.8 for more information)
   - Gas name [name of specie]
3. **Local gas properties**
   Two parameters of the gas in the volume have to be specified:
   - Gas constant R [Gas Constant Units]
   - Specific heat ratio κ [Ratio Units]

   For isothermal flow two additional parameters must be specified:
   - Initial temperature T_0 [Temperature Units]
   - Initial pressure p_0 [Pressure Units]

8.3.6. Governing Equations for Isothermal Flow

Pressure in **Two-Phase Volume** at isothermal flow is calculated from the continuity equation:

$$\dot{p}(t) = \frac{E_{2-ph}}{V(x)} \left\{ \sum_{i=1}^{n_1} \dot{Q}_i + \sum_{i=1}^{n_2} A_i \dot{x}_i - \sum_{j=1}^{l_1} \dot{Q}_j - \sum_{j=1}^{l_2} A_j \dot{x}_j \right\}, \tag{107}$$

$\dot{p}(t)$	pressure time derivative
$E_{2\text{-}ph}$	bulk modulus of two-phase (fluid-gas) mixture
$V(x)$	actual volume

$\dot{Q}_i$ and $\dot{Q}_j$	flow rates through i-th input and j-th output connections to non-piston-type elements
n_1 and l_1	number of hydraulic connections to non-piston-type elements on input and output ends
n_2 and l_2	number of hydraulic connections to piston-type elements on input and output ends
A_i and A_j	cross-sectional areas of piston-type elements connected to Volume by i-th input and j-th output connection, respectively
$\dot{x}_i$	velocity of piston-type element connected to Volume by i-th input connection
$\dot{x}_j$	the velocity of piston-type element connected to Volume by j-th output connection

Two-Phase Volume may have only hydraulic connections. Depending on the connected elements, these connections are divided into piston-type or non-piston-type. Piston-type connections imply the connection to the following elements: **Piston, SID Piston** (Piston group), **Plunger, RPD Pump** (Pump group), **Solenoid** Armature (Solenoid group) and any **Needle** element (Needle group). All other hydraulic connections to **Volume** are non-piston-type (to the elements of **Boundary, Line, Bend, Junction, Port, Valve, Throttle, Leakage, Orifice** and **Nozzle** groups).

Actual volume is calculated from the following equation:

$$V(x) = V_0 - \sum_{i=1}^{n_2} A_i x_i + \sum_{j=1}^{l_2} A_j x_j \ , \tag{108}$$

where V_0 is the initial volume and x_i and x_j are the displacements of piston-type elements connected to **Volume** by *i-th* input and *j-th* output connection, respectively.

Equation 108 shows that positive displacement of the connected piston-type element on input end will compress the volume, while positive piston displacement on output end will expand the volume. Obviously, if there are no hydraulic connections to piston-type elements neither on input nor on output end, the volume will remain constant.

8.3.6.1. Model of Gas Bubble Dynamics

Small Gas Fraction (Cavitation)

Basic assumptions:

- the model is based on the Rayleigh-Plesset theory of bubble dynamics
- fluid is assumed to contain a large number of tiny spherical bubbles
- bubbles can expand, contract or collapse depending on the pressure around them
- bubbles are assumed to be filled with an ideal gas

The fluid in the model is assumed to be a mixture of liquid and gas. The gas always exists in the liquid, e.g. in the form of a high number of very small bubbles. The mixture is assumed to be homogenous. However, the local gas content is still considered as one separate bubble. The behavior of such an individual bubble in the liquid is governed by the Rayleigh-Plesset equation of bubble dynamics without thermal effects[3]. The bubble

[3] Brennen, Christopher E. *Cavitation and Bubble Dynamics*. Oxford University Press, 1995.

expands, contracts or collapses depending on the pressure inside the bubble and the external pressure in the liquid. The variation of the radius of the bubbles determines the void fraction and thus influences the physical properties of the two-phase fluid such as the velocity of sound.

The basic problem in bubble dynamics is to determine the pressure, velocity and the motion of the bubble wall under the influence of the time-dependent pressure. The differential equations of bubble dynamics essentially describe how the radius of the cavity varies with time.

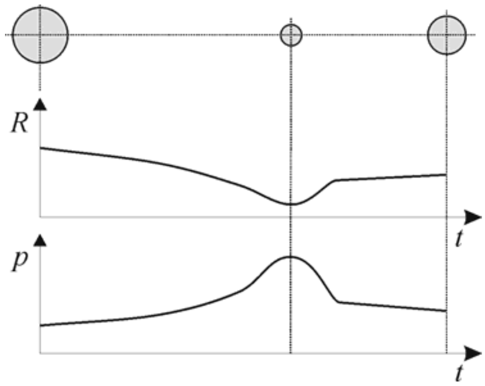


Figure 66: Contraction and Expansion of a Bubble

The equation of bubble dynamics in the incompressible fluid developed by Noltingk-Neppiras is

$$R\ddot{R}+\frac{3}{2}\dot{R}=\frac{1}{\rho}\left[\left(p_0+\frac{2\sigma}{R_0}\right)\left(\frac{R_0}{R}\right)^{2\pi}-\frac{2\sigma}{R}-p_\infty(t)\right]. \tag{109}$$

Bubble dynamics in the compressible fluid is governed by the Herring-Trilling equation:

$$R\ddot{R}\left(1-\frac{2\dot{R}}{a}\right)+\frac{3}{2}\dot{R}^2\left(1-\frac{4}{3}\frac{\dot{R}}{a}\right)=\frac{1}{\rho}\left[p(R)-p_\infty(t)+\frac{R}{a}\left(1-\frac{\dot{R}}{a}\right)\frac{dp_R}{dt}\right]. \tag{110}$$

Volumetric void fraction is defined by:

$$\beta_v=\left[\frac{1}{\frac{\beta_0}{1-\beta_0}\left(\frac{R_0}{R}\right)^3}+1\right]^{-1}. \tag{111}$$

The velocity of sound in the two-phase mixture can be expressed:

$$a_{2-ph}=\left[\sqrt{(1-\beta_v)\rho_{liquid}\left(\frac{1-\beta_v}{E_{liquid}}+\frac{\beta_v}{p}\right)}\right]^{-1}. \tag{112}$$

Two-phase mixture density is given by:

$$\rho_{2-ph} = (1-\beta_v)\rho_{liquid} . \tag{113}$$

Bulk modulus of the two-phase mixture can be obtained from:

$$E_{2-ph} = a_{2-ph}^2 \rho_{2-ph} . \tag{114}$$

Pressure inside bubble is calculated according to the following formula:

$$p_{b\_in} = \left(p + \frac{2\sigma}{R_0}\right)\left(\frac{R_0}{R}\right)^{3n} . \tag{115}$$

8.3.6.2. Fluid-Gas Mixture

Mass fraction of gas is calculated from the following equation:

$$\beta_m = \frac{m'(t)}{m_{total}(t)} . \tag{116}$$

β_m	mass fraction of gas
$m'(t)$	mass of gas in fluid-gas mixture
$m_{total}(t)$	total mass of fluid-gas mixture

Mass of gas is calculated according to the gas inflow into the Volume and fluid-gas mixture outflow from the Volume:

$$m'(t) = m'_0 + \sum m'_{inp} - \beta_{m_0} \sum m_{out} .$$

$m'(t)$	mass of gas in fluid-gas mixture
m'_0	mass of gas in Two-phase Volume at time t0
$\sum m'_{inp}$	mass inflow of gas into Volume
$\sum m_{out}$	mass outflow of fluid-gas mixture out of Volume
β_{m_0}	mass fraction of gas in Volume at time t0

Mass of liquid is calculated according to the fluid inflow into Volume and fluid-gas mixture outflow from the Volume:

$$m(t) = m_0 + \sum m_{inp} - (1-\beta_{m_0})\sum m_{out} .$$

$m(t)$	mass of fluid in fluid-gas mixture
m_0	mass of fluid in Two-phase Volume at time t0
$\sum m_{inp}$	mass inflow of fluid into Volume
$\sum m_{out}$	mass outflow of fluid-gas mixture out of Volume
β_{m_0}	mass fraction of gas in Volume at time t0

Total mass of fluid-gas mixture in Two-phase Volume is calculated as:

$$m_{total}(t) = m(t) + m'(t).$$

Gas inflow into **Two-Phase Volume** is calculated from the connected **Extended SAC Nozzle** element as backward flow. This calculation is performed only if a **Gas Pressure/Temperature Boundary** is connected on the other side of the Nozzle element.

Assuming isentropic gas fraction compression, density of gas at each state can be calculated from the gas law:

$$\rho_{gas} = \frac{p}{RT} \quad \text{or} \quad \rho_{gas} = \frac{p}{RT_0\left(\frac{p}{p_0}\right)^{\frac{\kappa-1}{\kappa}}}. \tag{117}$$

The left equation is used for the thermal fluid flow and the right equation – for isothermal fluid flow (as defined in **Calculation task** in **Simulation | Mode** menu). Note that compression/expansion of gas is in any case not isothermal but isentropic.
Velocity of sound a in the gas is determined using the formula:

$$a_{gas} = \sqrt{\frac{\kappa p}{\rho_{gas}}} = \sqrt{\kappa RT}. \tag{118}$$

Velocity of sound in the two-phase mixture is expressed as follows:

$$\frac{1}{a_{2-ph}^2} = \frac{\beta_v}{a_{gas}^2}\left[1 + (1-\beta_v)\left(\frac{\rho_{liquid}}{\rho_{gas}} - 1\right)\right] + \frac{1-\beta_v}{a_{liquid}^2}\left[1 + \beta_v\left(\frac{\rho_{gas}}{\rho_{liquid}} - 1\right)\right]. \tag{119}$$

Two-phase mixture density is defined by the equation:

$$\rho_{2-ph} = \rho_{liquid}(1-\beta_v) + \rho_{gas}\beta_v = \rho_{liquid} + \beta_v(\rho_{gas} - \rho_{liquid}), \tag{120}$$

where void fraction is given by: $$\beta_v = \frac{\frac{\beta_m}{\rho_{gas}}}{\frac{1-\beta_m}{\rho_{liquid}} + \frac{\beta_m}{\rho_{gas}}}. \tag{121}$$

For constant bulk modulus and thermal expansion coefficient of the liquid, its density is determined from the equation:

$$\rho_{liquid} = \rho_0 e^{\frac{1}{E_{liquid}}(p-p_0)} \rho_0 e^{-\alpha_{liquid}(T-T_0)} = \rho_0 e^{\frac{1}{E_{liquid}}(p-p_0)-\alpha_{liquid}(T-T_0)}. \tag{122}$$

Bulk modulus of the two-phase mixture is calculated from the formula:

$$E_{2-ph} = a_{2-ph}^2 \rho_{2-ph}. \tag{123}$$

$a_{2\text{-}ph}$	Velocity of sound in liquid-gas mixture
a_{liquid}	Velocity of sound in liquid
a_{gas}	Velocity of sound in gas
$E_{2\text{-}ph}$	Bulk modulus of liquid-gas mixture
E_{liquid}	Bulk modulus of liquid
p_{∞}	Pressure at infinity
p_0	Reference pressure in fluid
α_{liquid}	Thermal expansion coefficient of liquid
R	Gas constant
R_0	Bubble radius at reference pressure and temperature
T_0	Reference temperature in fluid
σ	Liquid surface tension
β_v	Volumetric void fraction of gas
β_m	Mass fraction of gas
κ	Specific heat ratio of gas
β_0	Void fraction of gas at reference pressure and temperature
$\rho_{2\text{-}ph}$	Density of liquid-gas mixture
ρ_{liquid}	Density of liquid
ρ_0	Liquid density at reference pressure and temperature
ρ_{gas}	Density of gas

8.3.7. Governing Equations for Thermal Flow

Pressure and temperature in **Two-phase Volume** at thermal flow is formally calculated from the same equations as in **Standard Volume** (refer to Section 8.1.7). However, here the density and bulk modulus of the two-phase (liquid-gas) mixture are used. These properties are calculated according to the equations 113 and 114 for the **Gas Bubble Dynamics** option and equations 120 and 123 for the **Fluid-gas Mixture** option, respectively.

For rigorous calculation, the specific heat capacity of the two-phase mixture has to be applied. However, as it is unknown, the specific heat capacity of liquid c is used instead. Thus, the equations governing fluid pressure and temperature have the form:

$$\dot{p}(t) = \frac{E_{2-ph}}{m}\left(\dot{m} - \rho_{2-ph}\dot{V}\right) + E_{2-ph}\alpha\dot{T}\ ,$$

$$\dot{T}(t) = \frac{1}{mc}\left[\sum\left(c_i T_i - cT\right)\dot{m}_i + p\left(\sum \dot{q}_i - \dot{V}\right) - \dot{Q}\right]. \tag{124}$$

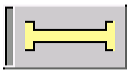
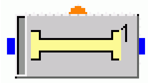
9. LINE

Note: General remark for all Line (duct/pipe/tube) elements:
Hydraulic diameter D_h is calculated from the formula:

$$D_h = \frac{4A_{nom}}{P_{nom}},$$

Where A_{nom} is the nominal cross-sectional flow area and P_{nom} is the inner perimeter of the line cross-section. For a circular line, hydraulic diameter is equal to the line diameter $(D_h = D)$.

9.1. d'Alembert Line

Element Name:	d'Alembert Line	
Element Icon:		
Definition:	This element serves to define a line (duct/pipe/tube). The solution of the line equation (without frictional losses) is derived by d'Alembert. Friction function has to be defined externally through empirical pressure-pulse damping.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Note: If local **variable fluid properties** are activated, local velocity of sound, fluid density and viscosity inside line will be calculated at the beginning of calculation according to initial pressure on input and output end of line. Otherwise (by default), **global fluid properties** will be used.
During calculation velocity of sound, fluid density and viscosity will remain constant. If set to variable, they will be averaged at calculation start.

Note: Usage of **empirical friction model** (pressure-pulse damping) implies the possible violation of mass conservation law.

Schematic of **d'Alembert Line** with expansions and contractions at its ends is shown in Figure 67.

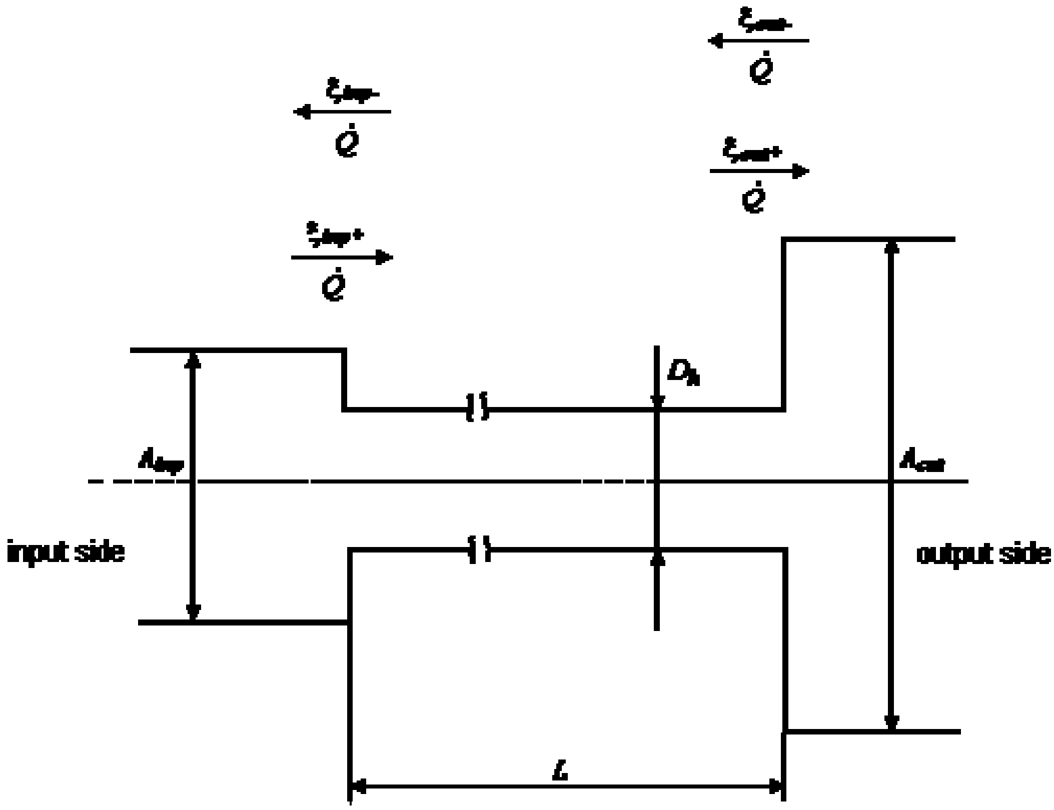


Figure 67: Geometry of d'Alembert Line

9.1.1. Input Parameters

Description of input data for the **d'Alembert Line**:

<table>
<tr><td>Element name:</td><td colspan="2">The name of the element is specified as default.</td></tr>
<tr><td>Fluid Properties</td><td colspan="2">Press this bar to specify the local fluid properties (refer to Section 8.1.4).</td></tr>
<tr><td>Line length:</td><td colspan="2">Specify length of line. It corresponds to L on Figure 67.</td></tr>
<tr><td>Hydraulic diameter:</td><td colspan="2">Specify hydraulic diameter of line. It corresponds to D_h in Figure 67.</td></tr>
<tr><td rowspan="3">Exponent of pressure-pulse damping:</td><td colspan="2">Specify coefficient of pressure-pulse damping β. Exponent β must be negative or in limiting case 0 (no damping).
The pressure-pulse damping is related to the propagation pressure disturbance from input to and output end of line. The following formula is an empirical function accounting for frictional losses in the line..
$$\Delta p_{out}(t) = \Delta p_{in}(t - \Delta t)e^{\beta L}$$
where:</td></tr>
<tr><td>Δp_{out}</td><td>Pressure disturbance on output end of line at time t.</td></tr>
<tr><td>$\Delta p_{in}(t-\Delta t)$</td><td>Pressure disturbance on input end at time $t-\Delta t$.</td></tr>
</table>

	$\Delta t(L)=L/a$	time a pressure wave needs to travel from line input to output end
	a	Velocity of sound in the fluid
	L	line length
	The damping coefficient β can be estimated from the following formula.4 $$\beta = -\frac{6.6}{D_h}\sqrt{\frac{\nu}{aL}},$$ where ν is kinematic viscosity of the fluid. This friction model is not exact from the physical point of view, but gives sufficient accuracy in many practical cases.	
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS:	Refer to Figure 67.	
Cross-sectional area at input end:	Specify cross-sectional area at input end A_{in}. Leaving default value 0 implies no expansion/contraction at input end. This input is active only if the **Expansions/Contractions at Connections** option is selected.	
Cross-sectional area at output end:	Specify cross-sectional areas at output end A_{out}. Leaving default value 0 implies no expansion/contraction at output end. This input is active only if the **Expansions/Contractions at Connections** option is selected.	
Positive flow resistance coefficient at input end:	Specify resistance coefficients ξ_{in+} for positive flow (from input end to output end) related to nominal area A_{nom} at sudden change of cross-sectional area on input end (refer to Figure 69 and Figure 70). This input is active only if the **Expansions/Contractions at Connections** option is selected.	
Positive flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output end (refer to Figure 69 and Figure 70). This input is active only if the **Expansions/Contractions at Connections** option is selected.	
Negative flow resistance coefficient at input end:	Specify resistance coefficient ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on input end. This input is active only if the **Expansions/Contractions at Connections** option is selected.	

4 B.E.Romig, R.D.Strunk and M.S.Weinert, *Perfomance Comparison of Unit Injector and Pump-Line-Nozzle Injection System*, SAE Paper 840274.

Negative flow resistance coefficient at output end:	Specify resistance coefficients ξ_{out-} for negative flow (from output end to input end) related to A_{nom} at sudden change of cross-sectional area on output end. This input is active only if the **Expansions/Contractions at Connections** option is selected.

ξ_{in+} and ξ_{in-} have to be defined only if A_{in} is specified (i.e. $A_{in} \neq A_{nom}$).
ξ_{out+} and ξ_{out-} have to be defined only if A_{out} is specified (i.e. $A_{out} \neq A_{nom}$).

Compliant walls:	This button activates the inclusion of wall compliance into calculation.
Wall thickness:	Enter thickness of line walls. This input is active only if the **Compliant walls** button is selected.
Young's modulus:	Enter Young's modulus of wall material. This input is active only if the **Compliant walls** button is selected.

The pressure wave propagates along the line with the velocity of sound:

$$a = \sqrt{\frac{E}{\rho}}, \tag{125}$$

where E is the modulus of elasticity and ρ is the fluid density.

The velocity of sound within the high-pressure lines is affected by the elasticity of line walls, which is encountered through the following equation5:

$$\frac{1}{E} = \frac{1}{E_{wall}} + \frac{D_h}{s}\frac{1}{E_{fluid}} \tag{126}$$

where:

s	wall thickness
E_{wall}	Young's modulus of line wall material
E_{fluid}	bulk modulus of the fluid

The line wall distensions cannot be considered in **d'Alembert Line** because the fluid flow is calculated only at line ends.

9.1.2. Initial Conditions

Path: **Element** | **Initial Conditions**

Description of initial conditions:

Flow Rate at x-input end:	Unit:	Volume Flow Units
Flow Rate at x-output end:	Unit:	Volume Flow Units

All initial values that are not specified in this dialog box are set to 0.

[5] Marcic, Milan. *Calculation of the Diesel Fuel Injection Parameters*. SAE Paper 952071.

9.1.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the button in front of modifiable parameters has to be checked. When modification is activated, the modification table has to be defined. Open it by pressing **Modify...** bar.

Modifiable Parameters:

exponent of pressure-pulse damping:	Unit:	---
cross-sectional area at input connection:	SI Unit:	m^2
resistance coefficient at input for positive flow	Unit:	---
resistance coefficient at input for negative flow	Unit:	---
cross-sectional area at output connection:	Unit:	m^2
resistance coefficient at output for positive flow	Unit:	---
resistance coefficient at output for negative flow	Unit:	---

9.1.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameters Dialog Box** of **d'Alembert Line** is shown in Figure 68.

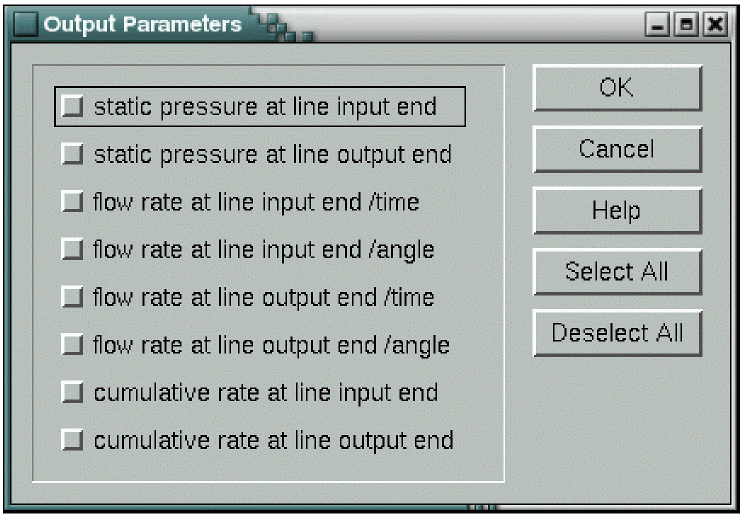


Figure 68: Output Parameters of d'Alembert Line Dialog

To activate output parameter, activate the check box in front of output parameter.

9.1.5. Estimation of Flow Resistance Coefficients

If better estimates are not available, flow resistance coefficients at Contractions/Expansions can be obtained from the following formulas.

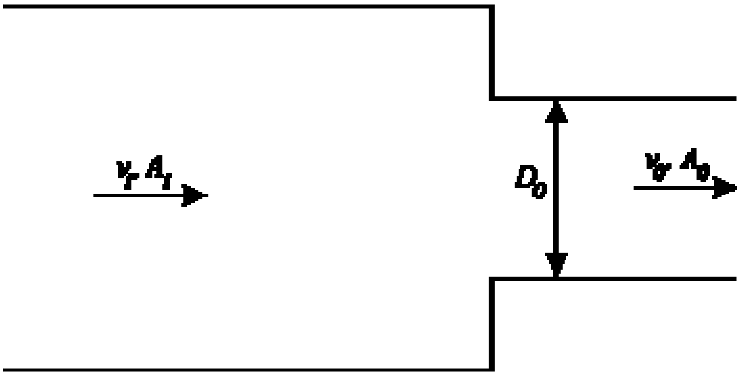


Figure 69: Sudden Contraction at Line End

At sudden contraction, flow resistance coefficient ξ can be estimated by calculating Reynolds number:

$$\mathrm{Re} = \frac{v_0 D_h}{\nu}, \qquad (127)$$

where:

$D_h = \frac{4A_{nom}}{P_{nom}}$ .... hydraulic diameter of line

v_o .... flow velocity

ν .... fluid viscosity

Re .... Reynolds number

i) if $10 < \mathrm{Re} \leq 104$ (turbulent flow in orifice) then ξ can be taken from Table 10 located in Chapter 19.1.

ii) if $1 < \mathrm{Re} < 6\text{-}7$ (laminar flow in orifice), the following formula can be used $\xi = \frac{30}{\mathrm{Re}}$

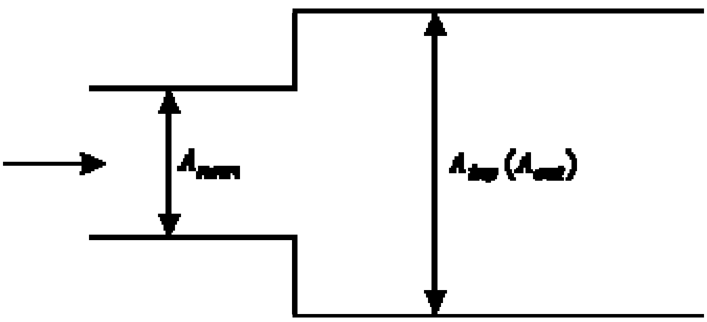


Figure 70: Sudden Expansion at Line end

At sudden expansion, flow resistance coefficient ξ can be estimated from Carnot formula:

$$\xi_{in} = \frac{1}{2}\left(1 - \frac{A_{nom}}{A_{in}}\right) \; ; \; \xi_{out} = \left(1 - \frac{A_{nom}}{A_{out}}\right)^2 \tag{128}$$

9.1.6. Basic Differential Equations

Differential equations of fluid flow in lines (pipes) are derived using the following assumptions:

- Elasticity of the line walls may be neglected compared with the compressibility of the fluid
- Temperature variation is small enough so that the viscosity may be considered to be constant
- Velocity and the variation of all dependent variables in the circumferential direction (Φ) is negligible due to rotational symmetry
- Flow is laminar or turbulent. Laminar flow implies that the Reynolds number is below 2300.

It is convenient to use cylindrical coordinates whose x-axis is identified with the center line of the pipe as shown in Figure 71. Let *r* be the coordinate in the radial direction and *t* denote time.

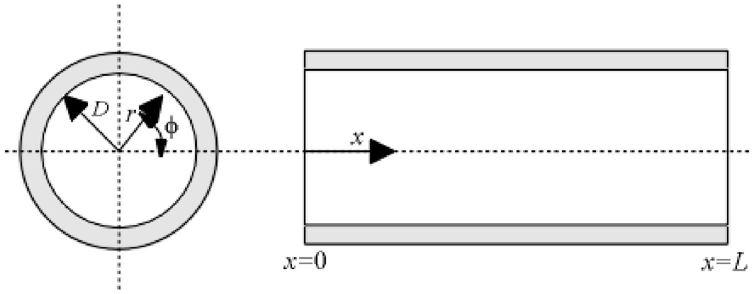


Figure 71: Pipe with Cylindrical Coordinates

The deviations of the velocity in the x and r-directions from their steady-state values are denoted by *v(x,r,t)* and *u(x,r,t),* respectively, and the deviation of the pressure by *p(x,r,t).* Let ρ designate the fluid density, μ the absolute (dynamic) viscosity.

The complete Navier-Stokes equations in cylindrical coordinates are given by Pai [9]. These equations are a result of **the law of conservation of momentum.** With the forgoing assumptions they are simplified and given in the following form.

Equation of motion in x-direction:

$$\rho\left[\frac{\partial v}{\partial t} + v\frac{\partial v}{\partial x} + u\frac{\partial v}{\partial r}\right] = -\frac{\partial p}{\partial x} + \mu\left[\frac{4}{3}\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial r^2} + \frac{1}{r}\frac{\partial v}{\partial r} + \frac{1}{3}\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial r} + \frac{u}{r}\right)\right] \tag{129}$$

Equation of motion in r-direction:

$$\rho\left[\frac{\partial u}{\partial t} + v\frac{\partial u}{\partial x} + u\frac{\partial u}{\partial r}\right] = -\frac{\partial p}{\partial r} + \mu\left[\frac{4}{3}\frac{\partial^2 u}{\partial r^2} + \frac{4}{3}\frac{1}{r}\frac{\partial u}{\partial r} - \frac{4}{3}\frac{u}{r^2} + \frac{\partial}{\partial x}\left(\frac{1}{3}\frac{\partial v}{\partial r} + \frac{\partial u}{\partial x}\right)\right] \tag{130}$$

where:

x, r	cylindrical coordinates
v, u	velocities in x and r direction
p	pressure
ρ	fluid density
μ	absolute (dynamic) viscosity
t	time

The following equation results from the mass conservation law. Let us consider a control volume with mass flows into and out of the volume. Accounting for the entire fluid mass and assuming the medium to be continuous, the mass storage rate has to be equal to the difference between incoming and outgoing mass flow rates.

Continuity equation has the general form:

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial u}{\partial r} + \rho \frac{u}{r} + \rho \frac{\partial v}{\partial x} + u \frac{\partial \rho}{\partial r} + v \frac{\partial \rho}{\partial x} = 0 . \tag{131}$$

The following assumptions are made:

- As $v >> u$, we neglect Equation 130. Eliminating of this equation implies that the pressure is constant across the cross-section of the tube and becomes a function only of x and t.
- In Equation 129 the nonlinear convective acceleration terms on the left-hand side may be neglected because $\frac{\partial v}{\partial t} >> v\frac{\partial v}{\partial x}$ and $\frac{\partial v}{\partial t} >> u\frac{\partial v}{\partial x}$.
- It can be shown that the only important viscous terms on the right hand side of Equation 129 are $\frac{\partial^2 v}{\partial r^2}$ and $\frac{1}{r}\frac{\partial v}{\partial r}$. The other viscous terms in this equation may be neglected.
- In Equation 131, the terms $u\frac{\partial \rho}{\partial r}$ and $v\frac{\partial \rho}{\partial x}$ may be neglected as the second order terms.

With the forgone assumptions, Equations 129 to 131 reduce to the following two differential equations:

$$\rho \frac{\partial v}{\partial t} + \frac{\partial p}{\partial x} = \mu \left[\frac{\partial^2 v}{\partial r^2} + \frac{1}{r}\frac{\partial v}{\partial r} \right]. \tag{132}$$

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial v}{\partial x} = 0 . \tag{133}$$

The system of Equations 132 and 133 has three unknowns. These are x coordinate of the infinitesimally small control volume, pressure and density.

Therefore three independent equations are required to solve them. The third equation is the equation of state. It may be derived from the linear acoustic theory:

$$\frac{\partial p}{\partial \rho} = a^2 = const. \tag{134}$$

where a is velocity of sound in the fluid.

Equation 134 is used in BOOST Hydsim for d'Alembert, Laplace, and Characteristics lines.

Using Equation 134, the first term of the continuity Equation 131 can be rewritten as:

$$\frac{\partial \rho}{\partial t} = \frac{\partial \rho}{\partial p}\frac{\partial p}{\partial t} = \frac{1}{a^2}\frac{\partial p}{\partial t}. \tag{135}$$

After simple transformations, Equations 132 and 133 reduce to:

$$\frac{\partial p}{\partial t} + \rho a^2 \frac{\partial v}{\partial x} = 0, \tag{136}$$

$$\frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{\partial v}{\partial t} = R, \tag{137}$$

where R is the friction function given by:

$$R = \frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial r^2} + \frac{1}{r}\frac{\partial v}{\partial r}\right]. \tag{138}$$

The above equations are known as Equations of Allievi [6].

9.1.7. D'Alembert Solution

Method of d´Alembert yields the solution of the Allievi equations without friction.

Considering system of Equations 136 and 137 in normal form and assuming R=0, we obtain:

$$\frac{d}{dt}(p + \rho a v) = 0 \text{ along characteristic line ch1 (refer to Figure 72)} \tag{139}$$

$$\frac{d}{dt}(p - \rho a v) = 0 \text{ along characteristic line ch2 (refer to Figure 72)} \tag{140}$$

This means that the expressions in the brackets are constant.

[6] Seifert, H. *Instationäre Strömungsvorgange in Rohrleitungen an Verbrennungskraftmaschinen*. Springer Verlag, Berlin 1962.

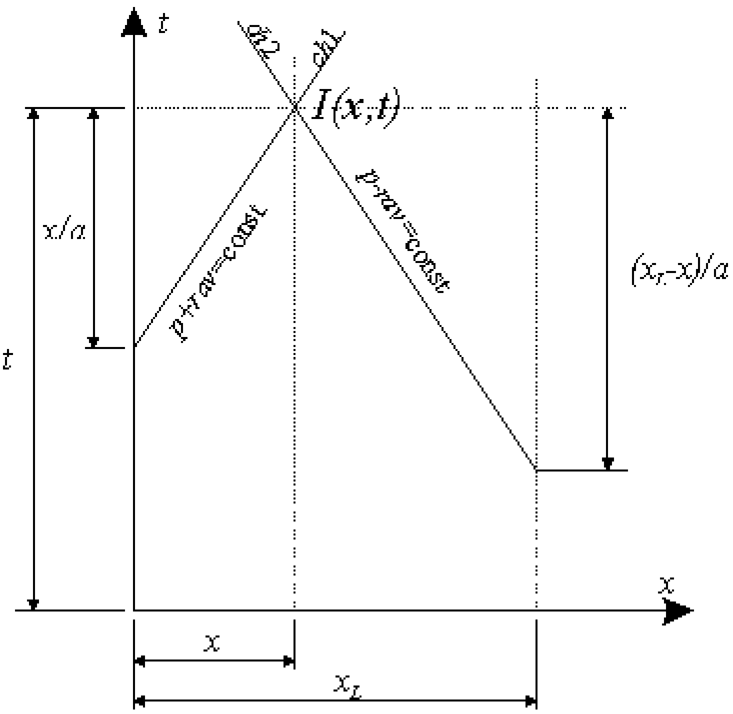


Figure 72: Characteristics at Line Ends

State variables at the point *I(x,t)* can be calculated from the coordinates of line ends (refer to Figure 72):

$$\text{ch1:}\quad p(x,t) + \rho a v(x,t) = p(0,t-\frac{x}{a}) + \rho a v(0,t-\frac{x}{a}), \tag{141}$$

$$\text{ch2:}\quad p(x,t) - \rho a v(x,t) = p(x_L,t-\frac{x_L - x}{a}) - \rho a v(x_L,t-\frac{x_L - x}{a}). \tag{142}$$

Rearranging the above equations, both state variables can be expressed in the form:

$$p(x,t) = \frac{1}{2}\left[[p+\rho a v](0,t-\frac{x}{a}) + [p-\rho a v](x_L,t-\frac{x_L - x}{a}) \right], \tag{143}$$

$$v(x,t) = \frac{1}{2\rho a}\left[[p+\rho a v](0,t-\frac{x}{a}) - [p-\rho a v](x_L,t-\frac{x_L - x}{a}) \right]. \tag{144}$$

Hence, flow velocity at line ends is calculated from:

$$v(0,t) = \frac{1}{\rho a} p(0,t) - \frac{1}{\rho a}[p-\rho a v](x_L,t-\frac{x_L}{a}), \tag{145}$$

$$v(x_L,t) = -\frac{1}{\rho a} p(x_L,t) + \frac{1}{\rho a}[p+\rho a v](0,t-\frac{x_L}{a}). \tag{146}$$

In the same way, pressures at line ends is given by:

$$p(0,t) = \rho a v(0,t) - [p-\rho a v](x_L,t-\frac{x_L}{a}), \tag{147}$$

$$p(x_L,t) = -\rho a v(x_L,t) + [p + \rho a v](0, t - \frac{x_L}{a}) . \quad (148)$$

In BOOST Hydsim, flow velocities v(0,t) and $v(x_L, t)$ are calculated inside the Line modules, while pressures $p(0,t)$ and $p(x_L, t)$ are taken from the connected elements (Volumes or Boundaries).

Damping models based on the Allievi equations without friction

To account for the frictional losses, some publications[7] introduce the damping function into the Allievi equations in the following way:

$$p(0,t) = \rho a v(0,t) + e^{\beta x_L} [p - \rho a v](x_L, t - \frac{x_L}{a}) , \quad (149)$$

$$p(x_L,t) = -\rho a v(x_L,t) + e^{\beta x_L} [p + \rho a v](0, t - \frac{x_L}{a}) . \quad (150)$$

Here β is a negative constant. It may be varied to obtain a suitable pressure curve matching the measurement. This model of frictional damping can work efficiently. However, it has several disadvantages:

- β has no physical meaning and has to be determined by experiment
- solution of the conservation laws is disturbed

The following semi-empirical function for β has been published:

$$\beta = -\frac{6.6}{D_h}\sqrt{\frac{\nu}{a x_L}} , \quad (151)$$

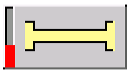
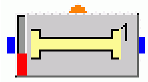
where:

D_h	inner diameter of line
ν	kinematic viscosity: $\nu = \frac{\mu}{\rho}$

With this simple model, satisfactory results can be achieved in many practical applications. However, one should be aware of the fact that the empirical friction model violates the mass conservation law. Hence **d'Alembert Line** model with friction cannot be used for accurate shock wave propagation analysis.

7 Romig, B.E., Strunk, R.D. and Weinert, M.S. *Performance Comparison of Unit Injector and Pump-Line-Nozzle System*. SAE Paper 840274.

9.2. Laplace Line

Element Name:	Laplace Line	
Element Icon:		
Definition:	This element serves to define a line of circular form (tube/pipe). The solution of the line equation with friction is derived by Kroller[8] using the Laplace transform. The non-stationary frictional losses are calculated by Melcher’s method9.	
Connecting pins:	Standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection are allowed.	

☞

Note: If local **variable fluid properties** are activated, local velocity of sound, fluid density and viscosity inside line will be calculated at the beginning of calculation according to initial pressure on input and output end of line. Otherwise (by default), **global fluid properties** will be used. **Laplace line** can work only with constant fluid properties. If global properties are set to variable, they will be averaged at the beginning of calculation and kept constant afterwards.

Note: If used with stationary flow (~constant pressure), **Melcher non-stationary friction model** may disturb the mass conservation law within **Laplace line**. Use frictionless model (option **No friction losses**) in this case.

Schematic of **Laplace Line** with expansions/contractions at the ends is shown in Figure 73.

[8] 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.

[9] K. Melcher. *Ein Reibungsmodell zur Berechnung von instationaeren Stroemungen in Rohrleitungen an Brennkraftmaschinen*, Bosch Tech. Berichte 4 (1974)7.

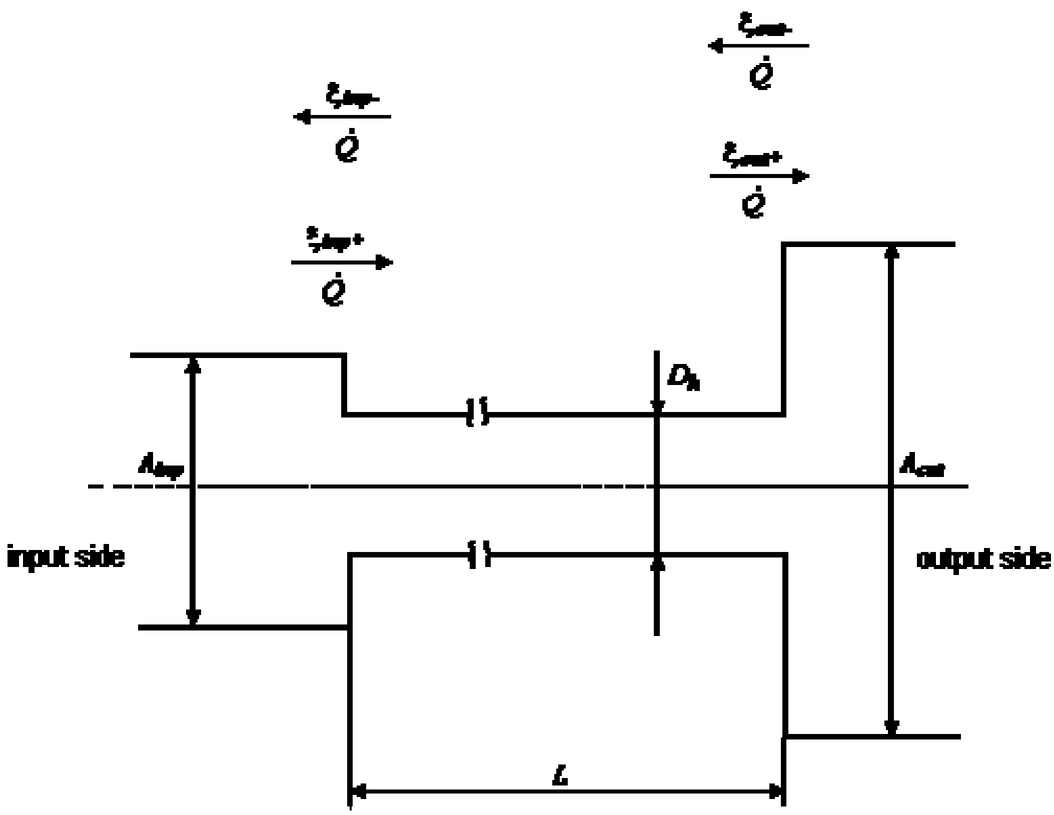


Figure 73: Geometry of Laplace Line

9.2.1. Input Parameters

Description of input data for the **Laplace Line**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Friction Losses	Press this bar to specify the model of friction losses to be used in the line. Friction models are explained in Section 9.2.5. **Note:** Melcher friciton model is not supported by the Variable step solver and in this case "no friction losses" model will be automatically taken .
Line length:	Specify length of line. It corresponds to *L* in Figure 73.
Hydraulic diameter:	Specify hydraulic diameter of line. It corresponds to D_h in Figure 73.
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS:	Refer to Figure 73 for line geometry.
Cross-sectional area at input end:	Specify cross-sectional area at input end A_{in}. Leaving default value 0 implies no expansions/contractions at input end. This input is active only if the **Expansions/Contractions at Connections** option is selected.

Cross-sectional area at output end:	Specify cross-sectional areas at output end A_{out}. Leaving default value 0 implies no expansions/contractions at input end. This input is active only if the **Expansions/Contractions at Connections** option is selected.
Positive flow resistance coefficient at input end:	Specify resistance coefficients ξ_{in+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional areas on input ends. This input is active only if the **Expansions/Contractions at Connections** option is selected.
Positive flow resistance coefficient at output end:	Specify resistance coefficients ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output ends. This input is active only if the **Expansions/Contractions at Connections** option is selected.
Negative flow resistance coefficient at input end:	Specify resistance coefficients ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on input end. This input is active only if the **Expansions/Contractions at Connections** option is selected.
Negative flow resistance coefficient at output end:	Specify resistance coefficients ξ_{out-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end. This input is active only if the **Expansions/Contractions at Connections** option is selected.

ξ_{in+} and ξ_{in-} have to be defined only if A_{in} is specified (i.e. $A_{in} \neq A_{nom}$).
ξ_{out+} and ξ_{out-} have to be defined only if A_{out} is specified (i.e. $A_{out} \neq A_{nom}$).

For the estimation of flow resistance coefficients, refer to Section 9.1.5.

Compliant Walls	This button activates the inclusion of wall compliance into calculation.
Wall thickness:	Enter thickness of line walls. This input is active only if the **Compliant walls** button is selected.
Young's modulus:	Enter Young's modulus of wall material. This input is active only if the **Compliant walls** button is selected.
The pressure wave propagates along the line with the velocity of sound given in Equation 79. The velocity of sound within the high-pressure lines is affected by the elasticity of the line walls. This is considered in the equation of the elasticity modulus (Equation 126).	

9.2.2. Initial Conditions

Description of initial conditions:

Flow Rate at x-input end:	Unit:	Volume Flow Units
Flow Rate at x-output end:	Unit:	Volume Flow Units

All initial values that are not specified in this dialog box are set to 0.

9.2.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, press the check button in front of modifiable parameter. When modification is activated, the modification table has to be opened by pressing **Modify...** bar.

Modifiable Parameters:

cross-sectional area at input connection	Unit:	m^2
resistance coefficient at input for positive flow	Unit:	---
resistance coefficient at input for negative flow	Unit:	---
cross-sectional area at output connection	Unit:	m^2
resistance coefficient at output for positive flow	Unit:	---
resistance coefficient at output for negative flow	Unit:	---

9.2.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameter Dialog Box** of **Laplace Line** is shown in Figure 74.

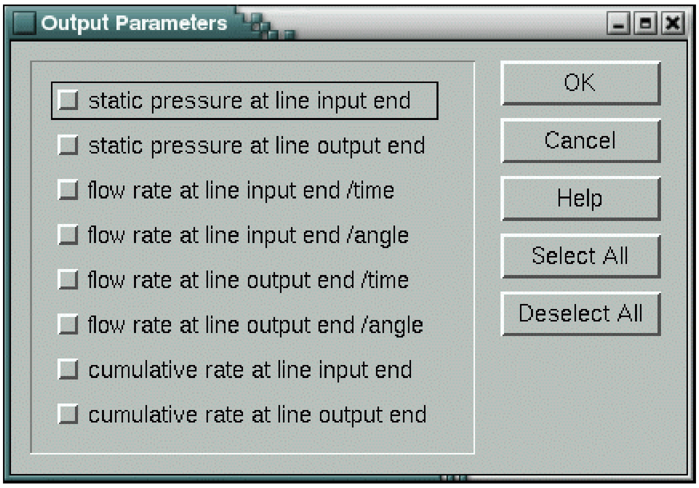


Figure 74: Output Parameters of Laplace Line Dialog

To activate output parameter, press the check button in front of output parameter.

9.2.5. Friction Losses

The calculation of one-dimensional unsteady liquid flow in a pipe is based on the laws of mass and momentum conservation (Equations 181 and 182). Friction force per unit mass ***R*** depends on the selected friction model. The following options are available:

none (no friction losses)

In this case there is no friction force per unit mass in liquid:

$$R = 0. \tag{152}$$

Melcher (non-stationary)

Melcher's method takes into account the non-stationary increase of friction. If A_f is the cross sectional area of the pipe, the averaged frictions force per unit mass for A_f is:

$$R = \frac{1}{A_f} \iint \upsilon \left(\frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) dydz\,. \tag{153}$$

The assumption is used that average friction force depends only on the shear stress on the wall. Melcher derived the following solution of friction function in the form:

$$R = \int_0^t \frac{\upsilon}{A_f} \frac{\partial v}{\partial t} D(t-\tau) d\tau\,, \tag{154}$$

where the damping function *D(t)* for circular pipe is given by:

$$D(t) = -4\pi \sum_1^\infty e^{-\frac{\omega_n^2 vt}{r^2}}\,. \tag{155}$$

Here ω_n are the roots of the Bessel function of zero order and r is the radius of the pipe cross section. In BOOST Hydsim, up to 20 roots can be used.

9.2.6. Theoretical Basis

The **Laplace Line** model is based on the Allievi Equations 136 and 137 (refer to Section 9.1.6). It may consider the non-stationary increase of friction by Melcher model. Due to the semi-analytical solution of the wave equation by Laplace transform[10] this model is computationally efficient (the fastest among the BOOST Hydsim line models) but less accurate.

The **Laplace Line** solution is based on the following assumptions:

[10] Kroller, M. *Efficient Computation of Mathematical Model for the Damping of Pressure Waves in Tubes of Circular Form.* Numerical Methods for Partial Differential Equations, 11, 41 – 60, 1995.

- Flow is weakly compressible, based on linear acoustic theory ($\frac{\partial p}{\partial \rho} = a^2 = const.$)
- Fluid flows within layers (velocity vectors are tangent to the streamlines)
- Flow pattern is an endless long straight cylinder of cross-sectional area f (refer to Figure 75)
- The friction function is independent of the x-coordinate, i.e. the fluid is assumed to consist of rigid shears (similar to Hagen-Poiseuille flow).

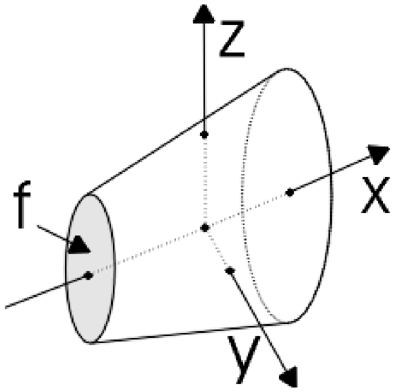


Figure 75: Flow Pattern as Long Straight Cylinder

With these assumptions, the Navier-Stokes equations reduce to:

$$\frac{\partial \upsilon}{\partial t} = \nu \left(\frac{\partial^2 \upsilon}{\partial y^2} + \frac{\partial^2 \upsilon}{\partial z^2} \right), \tag{156}$$

where υ is flow velocity and ν is kinematic viscosity.

In Equation 156, the expression on the right side represents the frictional force per unit mass. For Melcher non-stationary friction model, this averaged friction force is:

$$R = \frac{1}{f} \iint \nu \left(\frac{\partial^2 \upsilon}{\partial y^2} + \frac{\partial^2 \upsilon}{\partial z^2} \right) dydz , \tag{157}$$

where f is the cross sectional area of the pipe (refer to Figure 75).

In the work of Melcher[11], the solution of the differential Equation 157 is provided. It is based on the assumption that the average friction force depends only on the shear stress on the wall (formula of Green, Hagen-Poiseuille flow). Melcher derived the solution of friction function in the form of Equation 154 (refer to Section 9.2.5 for details).

[11] K. Melcher. *Ein Reibungsmodell zur Berechnung von instationaeren Stroemungen in Rohrleitungen an Brennkraftmaschinen*, Bosch Tech. Berichte 4 (1974)7.

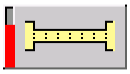
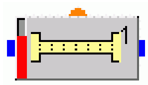
Hence, for Melcher friction model the equations of Allievi with constant cross-sectional area f and velocity υ have the form:

$$\text{(mass)} \quad \frac{\partial p}{\partial t} + a^2 \rho \frac{\partial \upsilon}{\partial x} = 0, \tag{158}$$

$$\text{(momentum)} \quad \frac{\partial \upsilon}{\partial t} + \frac{1}{\rho}\frac{\partial p}{\partial x} = \int_0^t \frac{\nu}{f}\frac{\partial \upsilon}{\partial t} D(t-\tau)\, d\tau . \tag{159}$$

Without friction losses ($\boldsymbol{R}=0$) the right hand side of momentum Equation 159 vanishes to zero.

9.3. Characteristics Line

Element Name:	Characteristics Line	
Element Icon:		
Definition:	This element serves to define a line (tube/pipe). The solution of the flow equation is obtained by the method of characteristics. For integration, the **Predictor – Corrector** of pure **Predictor scheme** is used. The non-stationary frictional losses are calculated by Melcher's method12.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection are allowed.	

☞ **Note:** Currently the Variable-step solver does not support this element.

☞ **Note:** If local **variable fluid properties** are activated, local velocity of sound, fluid density and viscosity inside line will be calculated at the beginning of calculation according to initial pressure on input and output end of line. Otherwise (by default), **global fluid properties** will be used. If global properties are set to variable, they will be averaged at calculation start.

Note: **Characteristics line** with **Melcher's non-stationary friction model** always satisfies the mass and momentum conservation laws.

[12] Melcher, K. *Ein Reibungsmodell zur Berechnung von instationären Strömungen in Rohrleitungen an Brennkraftmaschinen,* Bosch Tech Berichte 4 (1974)7.

Schematic of **Characteristics Line** with expansions/contractions at the ends is shown in Figure 76.

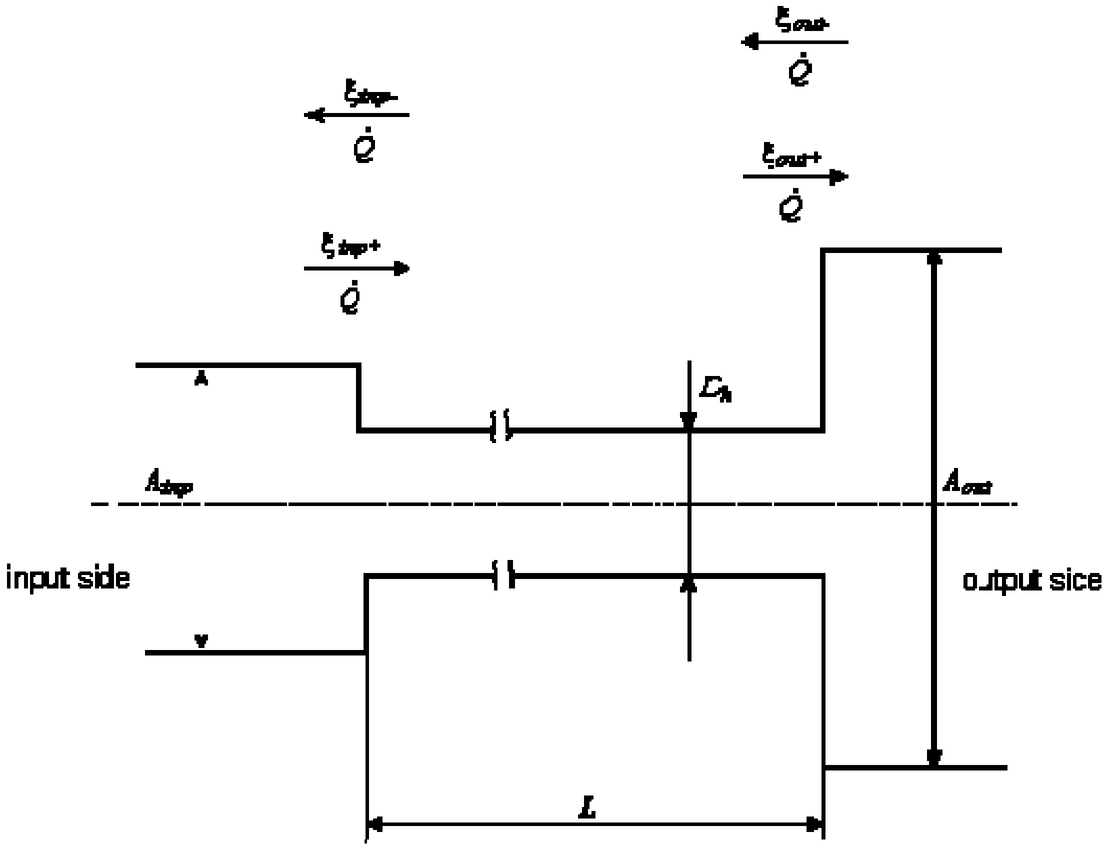


Figure 76: Geometry of Characteristics Line

9.3.1. Input Parameters

Description of input data for the **Characteristics Line**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Line length:	Specify length of line. It corresponds to *L* in Figure 76.
Hydraulic diameter:	Specify hydraulic diameter of line. It corresponds to D_h in Figure 76.
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS:	Refer to Figure 76 for line geometry.
Cross-sectional area at input end:	Specify cross-sectional areas at input end *Ain.* Leaving default value 0 implies no extension/contraction at input end. This input is active only if the **Expansions/Contractions at Connections** button is selected.

Cross-sectional area at output end:	Specify cross-sectional areas at output end A_{out}. Leaving default value 0 implies no extension/contraction at output end. This input is active only if the **Expansions/Contractions at Connections** button is selected.
Positive flow resistance coefficient at input end:	Specify resistance coefficient ξ_{in+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on input end. This input is active only if the **Expansions/Contractions at Connections** button is selected.
Positive flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output end. This input is active only if the **Expansions/Contractions at Connections** button is selected.
Negative flow resistance coefficient at input end:	Specify resistance coefficients ξ_{in-} for negative flow (from output end to input end) related to area A_h at sudden change of cross-sectional area on input end. This input is active only if the **Expansions/Contractions at Connections** button is selected.
Negative flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end. This input is active only if the **Expansions/Contractions at Connections** button is selected.

ξ_{in+} and ξ_{in-} have to be defined only if A_{in} is specified (i.e. $A_{in} \neq A_{nom}$).
ξ_{out+} and ξ_{out-} have to be defined only if A_{out} is specified (i.e. $A_{out} \neq A_{nom}$).

For the estimation of flow resistance coefficients, refer to Section 9.1.5.

Compliant Walls	This button activates the inclusion of wall compliance into calculation.
Wall thickness:	Enter thickness of line walls. This input is active only if the **Compliant walls** button is selected.
Young's modulus	Enter Young's modulus of wall material. This input is active only if the **Compliant walls** button is selected.
Poisson's ratio:	Enter Poisson's ratio of wall material. This input is active only if the **Compliant walls** button is selected. Pressure wave propagates along the line with the velocity of sound calculated from Equation 79. The velocity of sound in high-pressure lines is influenced by the elasticity of line walls. This effect is considered in the corrected bulk modulus given by Equation 126.

PREDICTOR MODEL (EMPIRICAL damping):	
Exponent of pressure-pulse damping:	Specify coefficient of pressure-pulse damping ζ**.** Exponent ζ must be negative or in limiting case 0 (no damping). For the **Predictor Model,** characteristics are calculated only at the line ends. This model has an exact solution if no frictional flow resistance is considered in the line and corresponds to the d'Alembert's solution of the wave equation (refer to **d'Alembert Line).** An empirical function for the flow resistance (damping) is applied to the analytical solution of the loss-free wave equation. This means that the resulting model is not a solution of the wave equation anymore. Thus, the conservation laws of mass and momentum are not fulfilled. The empirical function for the frictional losses is given by the following equation: $$\Delta p_{out}(t) = \Delta p_{in}(t - \Delta t)\, e^{\zeta L},$$ where: $\Delta p_{out}(t)$ is the pressure disturbance on output end of line at time t $\Delta p_{in}(t - \Delta t)$ is the pressure disturbance on input end of line at time $(t - \Delta t)$ Δt is the time a pressure wave needs to travel from line input to output end. The damping coefficient ζ is 0 if no frictional losses are considered and is negative if the losses are encountered. Keep in mind that this formula is not exact from the physical point of view, but provides acceptable results in most practical cases. The damping coefficient ζ can be estimated from the following formula13. $$\zeta = -\frac{6.6}{D_h}\sqrt{\frac{\nu}{aL}}$$ where: ν kinematic viscosity of the fluid a velocity of sound in the fluid This input is active only if the **Predictor Model** button is selected.
PREDICTOR - CORRECTOR MODEL:	

[13] Romig, B.E:, Strunk. and R.D. and Winert, M.S. *Performance Comparison of Unit Injector and Pump Line Nozzle System*, SAE Paper 840274.

Corrector step (discrete time interval):	Enter Corrector Step N_L (number f time steps after which damping has to be recalculated). $N_L = 1$ No Predictor step is required (pure Corrector model). $N_L \geq 2$ The characteristics are calculated from the "Discrete Characteristics Model". In this case, every N_L-*th* time-step is the Corrector step at which the damping function is recalculated for the Predictor model. This is used until the next Corrector step. This input is active only if the **Predictor-Corrector Model** button is selected.

☞ **Note:** N_L should not be larger than 5 in order to maintain numerical accuracy.

	For the **Predictor-Corrector** Model, the wave equation including a frictional function is solved by the method of characteristics. Thus, the line is split into discrete sections where the characteristics originate. The length of the discrete line sections is calculated by the equation of characteristics: $$\Delta x = 2aN_L\Delta t$$ As the numerical integration of e.g. diesel injection system requires rather small time steps, high numbers of discrete line sections *nx* might result from this equation: $$n_x = \frac{L}{\Delta x}$$ This may lead to very long calculation times. With the **Predictor-Corrector** Model, it is possible to choose individual (longer) time steps for the Line calculation. The input parameter N_L (length of the discrete line interval) defines for how many times the actual time step for the line calculation exceeds the system time step specified in the **Calculation Control** dialog. For the extrapolation between the system time step and line time step, the Predictor model is used with damping coefficient ζ calculated from the previous Corrector step. Friction function of **Characteristics Line** (circular tube) is calculated from Equation 154 [Melcher], where: $$D(t) = -4\pi\sum_{n=1}^{M} e^{\frac{-\omega_n \nu t}{R^2}},$$ where R is the tube radius, ω_n are the roots of Bessel function of zero order, ν is the kinematic viscosity and M is number of Bessel function roots to be considered.

Number of roots of Bessel function of order zero:	Enter number of roots of Bessel function of zero order (to be used in the calculation of damping). Currently maximum 20 roots allowed. This input is active only if the **Predictor-Corrector Model** option is selected.
Cross-section distance from input end (optional)	For the **Predictor/Corrector** Model, the Line can be divided into up to five sections for which the line pressure distribution can be calculated. In this way, up to five cross-sections can be defined in the input dialog by specifying their distance from the input side of line.

9.3.2. Initial Conditions

Initial Conditions of the **Characteristics Line** are identical to the **Initial Conditions** of **d'Alembert Line** and **Laplace Line.** Refer to Section 9.1.2 or 9.2.2 for more information.

9.3.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

exponent of pressure-pulse damping	SI Unit:	---
cross-sectional area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-sectional area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---

9.3.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameter Dialog Box** of **Characteristics Line** is shown in Figure 77.

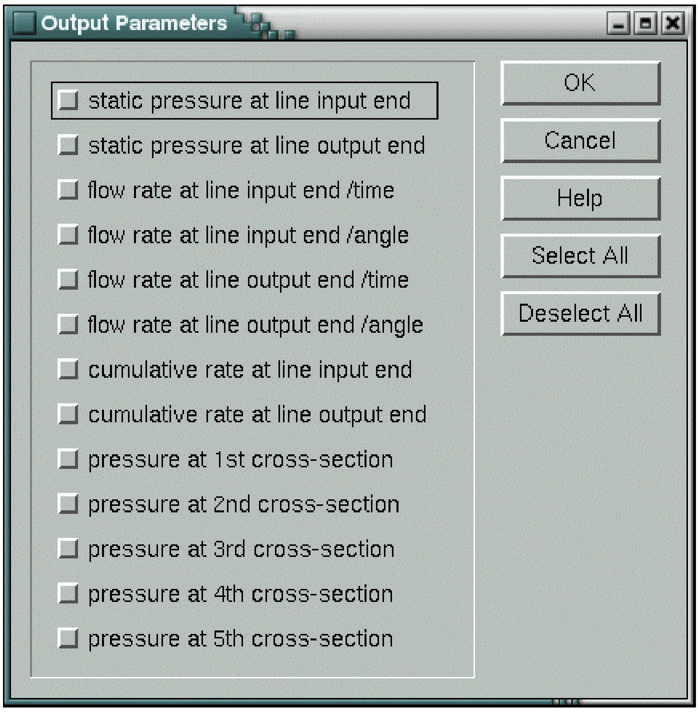


Figure 77: Output Parameters of Characteristics Line Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

For the pressures at cross-sections, the pressure at the *i-th* cross-section will be calculated only if this cross-section is defined in the input dialog (by specifying its distance from line input end).

9.3.5. Equations of Fluid Flow

The **Characteristics Line** is solved by the method of characteristics14.

Allievi Equations 136 and 137 can be transformed into the so-called "normal form" as given below:

$$\frac{1}{2\rho a}\left[\frac{\partial p}{\partial t}+a\frac{\partial p}{\partial x}\right]+\frac{1}{2}\left[\frac{\partial \upsilon}{\partial t}+a\frac{\partial \upsilon}{\partial x}\right]=\frac{1}{2}R\,,$$

$$\frac{1}{-2\rho a}\left[\frac{\partial p}{\partial t}-a\frac{\partial p}{\partial x}\right]+\frac{1}{2}\left[\frac{\partial \upsilon}{\partial t}-a\frac{\partial \upsilon}{\partial x}\right]=\frac{1}{2}R\,. \tag{158}$$

Here we have all derivatives in the form:

$$\frac{\partial p}{\partial t}\pm a\frac{\partial p}{\partial x}\;,\;\frac{\partial \upsilon}{\partial t}\pm a\frac{\partial \upsilon}{\partial x}\,, \tag{159}$$

[14] Roe, P.L. *Characteristic-based schemes for the Euler equations*. Annular Review Fluid Mech., 18, 1986. pp. 337-365.

where $\pm a$ are the eigenvalues of the system. Denoting $\lambda_i = \pm a$, the eigenvalues problems of Equations 158 can be defined as:

$$\frac{\partial p}{\partial t} + \lambda_i \frac{\partial p}{\partial x} \quad ; \quad \frac{\partial \upsilon}{\partial t} + \lambda_i \frac{\partial \upsilon}{\partial x} \quad ; \ \text{i=1,2.} \tag{160}$$

Eigenvalues are called the characteristic directions.

The solutions of the differential Equation 160 are given in the form:

$$\frac{dx}{dt} = \lambda_i \,. \tag{161}$$

These solutions, called the characteristics, are the following:

$$\frac{dx}{dt} = \lambda_1 = +a \Rightarrow x = a \cdot t + \text{const.} \quad \text{(=characteristic "ch1")} \tag{162}$$

$$\frac{dx}{dt} = \lambda_2 = -a \Rightarrow x = -a \cdot t + \text{const.} \quad \text{(=characteristic "ch2")} \tag{163}$$

Introducing the above solutions into the system of Allievi equations in ‘normal form’ (Equation 158), we obtain the system containing derivatives only along the characteristics:

$$\frac{\partial p}{\partial t} + \lambda_1 \frac{\partial p}{\partial x} = \frac{\partial p}{\partial t} + \frac{dx}{dt}\frac{\partial p}{\partial x} = \frac{dp}{dt} \ \text{ along "ch1",} \tag{164}$$

$$\frac{\partial p}{\partial t} + \lambda_2 \frac{\partial p}{\partial x} = \frac{\partial p}{\partial t} + \frac{dx}{dt}\frac{\partial p}{\partial x} = \frac{dp}{dt} \ \text{ along "ch2",} \tag{165}$$

$$\frac{\partial \upsilon}{\partial t} + \lambda_1 \frac{\partial \upsilon}{\partial x} = \frac{\partial \upsilon}{\partial t} + \frac{dx}{dt}\frac{\partial \upsilon}{\partial x} = \frac{d\upsilon}{dt} \ \text{ along "ch1",} \tag{166}$$

$$\frac{\partial \upsilon}{\partial t} + \lambda_2 \frac{\partial \upsilon}{\partial x} = \frac{\partial \upsilon}{\partial t} + \frac{dx}{dt}\frac{\partial \upsilon}{\partial x} = \frac{d\upsilon}{dt} \ \text{ along "ch2".} \tag{167}$$

Thus, Equation 158 can be rewritten as follows:

$$\frac{1}{2\rho a}\left[\frac{dp}{dt}_{ch1}\right] + \frac{1}{2}\left[\frac{d\upsilon}{dt}_{ch1}\right] = \frac{1}{2}R \,,$$

$$-\frac{1}{2\rho a}\left[\frac{dp}{dt}_{ch2}\right] + \frac{1}{2}\left[\frac{d\upsilon}{dt}_{ch2}\right] = \frac{1}{2}R \,. \tag{168}$$

Figure 78 represents numerical approximation of derivatives in Equation 168.

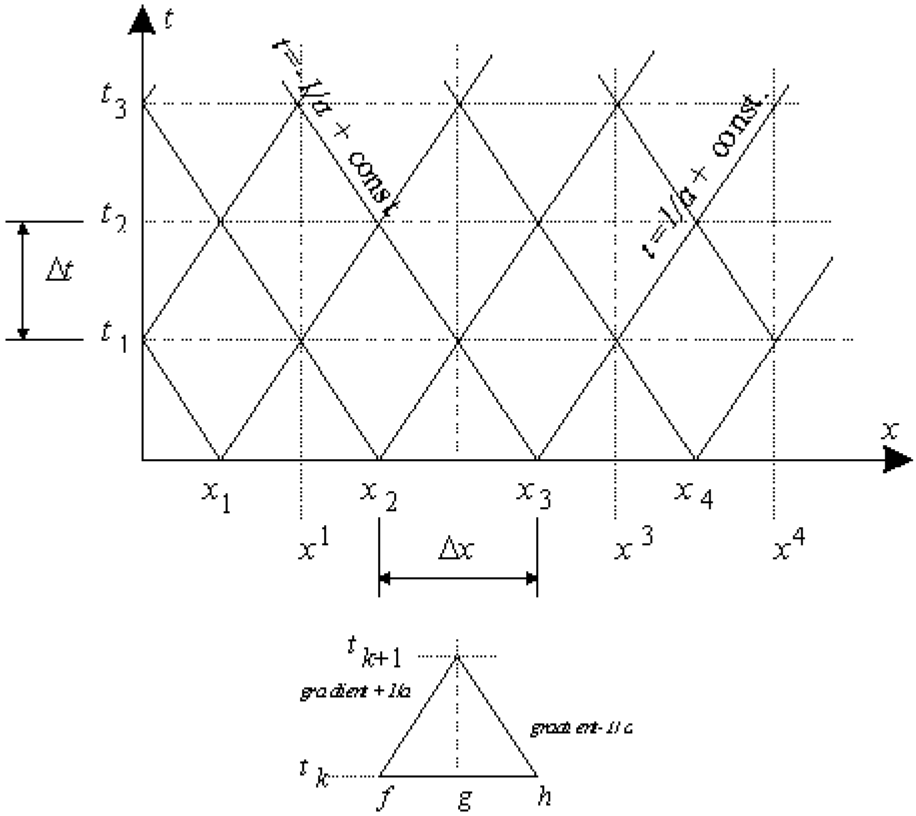


Figure 78: Numerical Approximation of Derivatives

According to the characteristics, the grid points x_i , t_k are related by the formula:

$$\Delta t = \frac{1}{a}\frac{\Delta x}{2}. \tag{169}$$

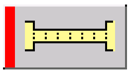
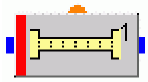
Using the previous approximation, the system of Equations 168 reduces to a simple algebraic form:

$$\frac{1}{2\rho a}\frac{p(g,t_{k+1})-p(f,t_k)}{\Delta t}+\frac{1}{2}\frac{\upsilon(g,t_{k+1})-\upsilon(f,t_k)}{\Delta t}-\frac{1}{2}R=0,$$

$$-\frac{1}{2\rho a}\frac{p(g,t_{k+1})-p(h,t_k)}{\Delta t}+\frac{1}{2}\frac{\upsilon(g,t_{k+1})-\upsilon(h,t_k)}{\Delta t}-\frac{1}{2}R=0. \tag{170}$$

Friction function R is calculated by Melcher's method as described in Section 9.2.5.

9.4. Godunov Line

Element Name:	Godunov Line
Element Icon:	
Definition:	This element serves to define a line (tube/pipe). The solution of the flow equation is obtained by Godunov method. Frictional losses can be switched off or calculated by various methods.
Connecting pins:	Standard pins: 2 (hydraulic) Special pins: 0 wire pins: 1 Only one input and one output connection have to be specified. 

Note: Currently the Variable-step solver does not support this element.

Note: **Godunov line** model is a complex 1st order fluid flow model based on the Godunov solution of the Riemann problem. It works with strongly variable fluid properties. The solution may lead to long calculation time.

Note: Friction in **Godunov line** can be modeled by **Melcher's non-stationary friction method, Colebrook-White equation** or using the **fixed friction factor Godunov line** can be used without friction losses, too. In all cases it satisfies the mass and momentum conservation laws.

Schematic of **Godunov Line** with expansions/contractions at the ends is shown in Figure 79.

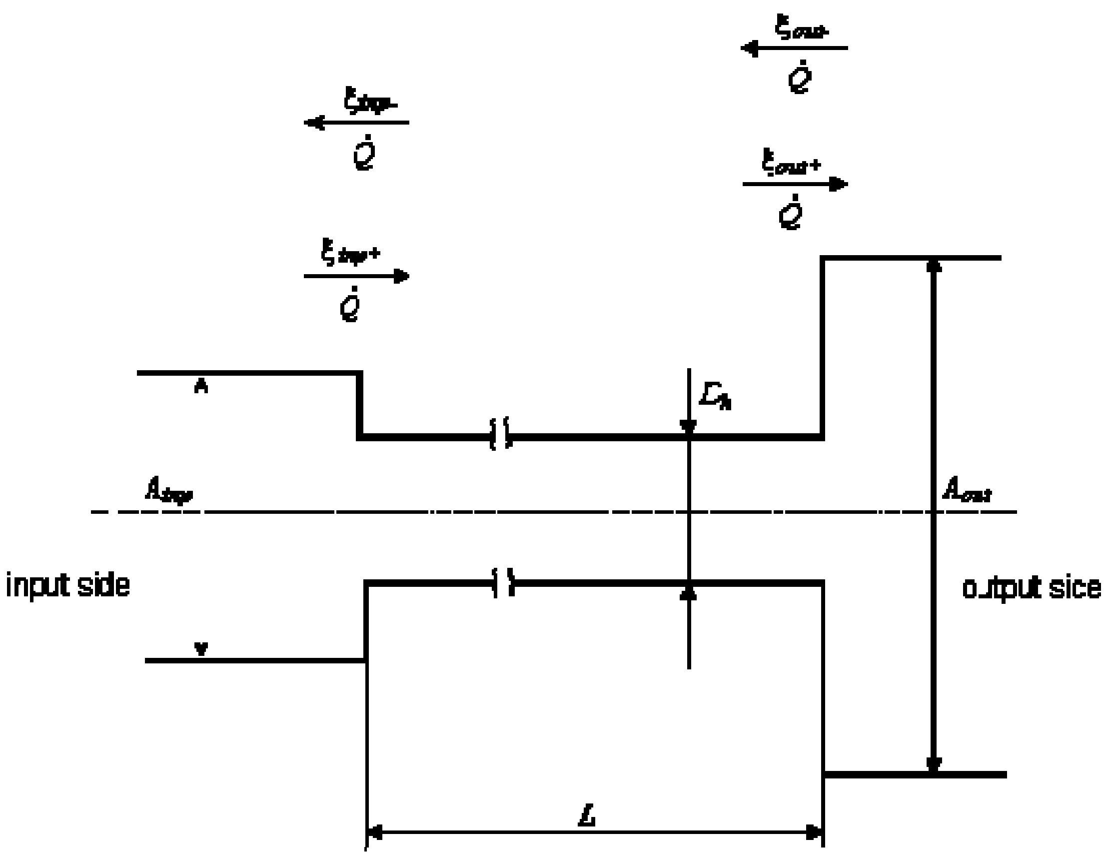


Figure 79: Geometry of Godunov Line

9.4.1. Input Parameters

Description of input data for the **Godunov Line**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Friction Losses	Press this bar to specify the model of friction losses to be used in the line. Friction models are explained in Section 9.2.5.
Line length:	Specify length of line. It corresponds to *L* in Figure 79.
Hydraulic diameter:	Specify hydraulic diameter of line. It corresponds to D_h in Figure 79.
Number of line cells:	BOOST Hydsim allows the user to define the number of cells (line divisions). According to it, the time step to cell length ratio is determined at calculation start using the specified initial conditions. Applying Courant-Friedrichs-Lewy (CFL) criterion, this ratio is checked at every time calculation step. CFL criterion is explained in Section 9.4.8.1. If not defined or switched off, number of cells is calculated internally.

EXPANSIONS/ CONTRACTIONS AT CONNECTIONS:	Refer to Figure 79 for more information.
Cross-sectional area at input end:	Specify cross-sectional area at input end A_{in}. Leaving default value 0 implies no extension/contraction at input end.
Cross-sectional area at output end	Specify cross-sectional areas at output end A_{out} (refer to Figure 79 for more information). Leaving default value 0 implies no extension/contraction at input end.
Positive flow resistance coefficient at input end:	Specify resistance coefficient ξ_{in+} for positive flow (from input end to output end) related to area *Anom* at sudden change of cross-sectional area on input end.
Positive flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output ends.
Negative flow resistance coefficient at input end:	Specify resistance coefficients ξ_{in-} for negative flow (from output end to input end) related to A_{nom} at sudden change of cross-sectional area on input end.
Negative flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to A_{nom} at sudden changes of cross-sectional areas on output ends.

ξ_{in+} and ξ_{in-} have to be defined only if Ain is specified (i.e. $A_{in} \neq A_{nom}$).
ξ_{out+} and ξ_{out-} have to be defined only if $Aout$ is specified (i.e. $A_{out} \neq A_{nom}$).

For the estimation of flow resistance coefficients, refer to Section 9.1.5.

Compliant Walls	This button activates the inclusion of wall compliance into calculation.
Wall thickness:	Enter thickness of line walls *s*. This input is active only if the **Compliant walls** button is selected.
Young's modulus:	Enter Young's modulus of wall material E_{wall}. This input is active only if the **Compliant walls** button is selected.
Poisson's ratio:	Enter Poisson's ratio of wall material. This input is active only if the **Compliant walls** button is selected.

The pressure wave propagates along the line with the velocity of sound as given in the equation:

$$a = \sqrt{\frac{E}{\rho}}, \tag{171}$$

where:

a .... velocity of sound in the fluid

ρ .... fluid density

E .... resultant elasticity modulus

<table>
<tr><td colspan="2">The velocity of sound within the high-pressure lines is also affected by the elasticity of the line walls. This is considered in the equation of the resultant elasticity modulus:
$$\frac{1}{E} = \frac{1}{E_{wall}} + \frac{D_h}{s} \frac{1}{E_{fluid}}, \qquad (172)$$
where:
s .... wall thickness
E_{wall} .... Young's modulus of wall material
E_{fluid} .... bulk modulus of the fluid
The line wall distensions are not considered in Godunov line model (i.e. D_h = const.).</td></tr>
<tr><td>Cross-sectional distance from input end:</td><td>Define up to 10 line cross-sections by specifying their distance from line input end. For these positions, BOOST Hydsim will calculate pressure and flow rate. This data can be stored on output if activated as output parameters.</td></tr>
</table>

9.4.2. Initial Conditions

Initial Conditions of **Godunov Line** are identical to the **Initial Conditions** of **d'Alembert Line** model and **Laplace Line** model (refer to Section 9.1.2).

9.4.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

hydraulic diameter	SI Unit:	m
wall thickness	SI Unit:	m
cross-sectional area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-sectional area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---
absolute roughness	SI Unit:	m
fixed friction factor	SI Unit:	---
bending radius	SI Unit:	m

9.4.4. Output Parameters

Path: **Element** | **Store Results**

The **Output Parameter Dialog Box** of **Godunov Line** is shown in Figure 80.

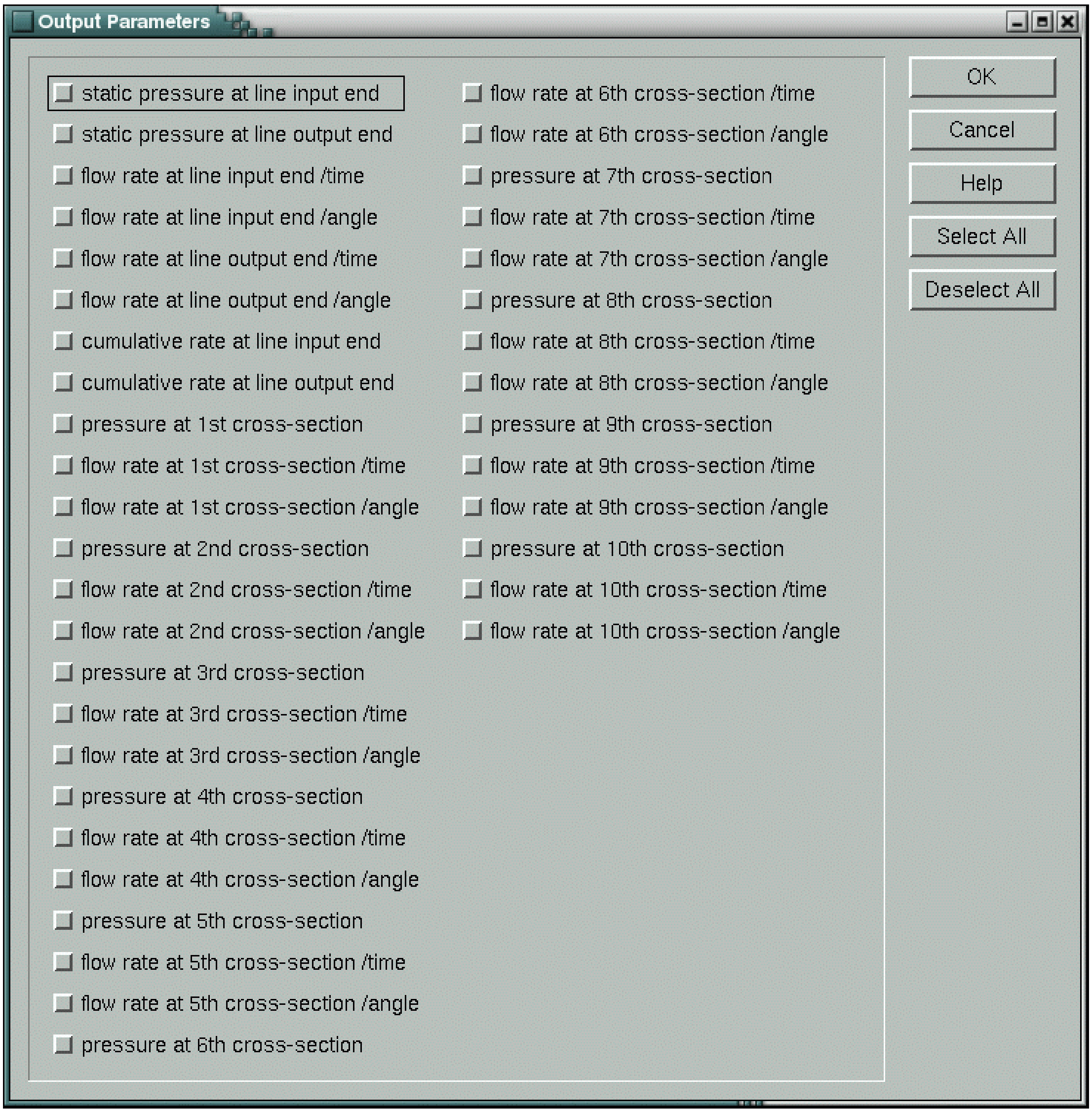


Figure 80: Output Parameters for Godunov Line Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

9.4.5. Friction Losses

The calculation of one-dimensional unsteady liquid flow in a pipe is based on the laws of mass and momentum conservation (Equations 181 and 182). Friction force per unit mass $\boldsymbol{R}$ depends on the selected friction loss model. The following options are available:

<u>none (no friction losses)</u>

In this case there is no friction force per unit mass in liquid:

$$R = 0. \tag{173}$$

Melcher (non-stationary)

Melcher's method takes into account the non-stationary increase of friction. If A_f is the cross sectional area of the pipe, the averaged frictions force per unit mass for A_f is:

$$R = \frac{1}{A_f} \iint \upsilon \left(\frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) dydz \,. \tag{174}$$

The assumption is used that average friction force depends only on the shear stress on the wall. Melcher derived the following solution of friction function in the form:

$$R = \int_0^t \frac{\upsilon}{A_f} \frac{\partial v}{\partial t} D(t - \tau) d\tau \,, \tag{175}$$

where the damping function *D(t)* for circular pipe is given by:

$$D(t) = -4\pi \sum_1^\infty e^{-\frac{\omega_n^2 vt}{r^2}} \,. \tag{176}$$

Here ω_n are the roots of the Bessel function of zero order and r is the radius of the pipe cross section. For details refer to Section 9.2.5.

Colebrook-White Equation

Friction force per unit mass R is complex function of the system geometry, fluid properties and flow rate. In most engineering flows, the heat loss is roughly proportional to the square of the flow rate (fully developed pipe flow). This observation leads to the Darcy-Weisbach equation for heat loss due to friction:

$$R = -\frac{1}{2} \frac{f}{D} v|v| \,, \tag{177}$$

where f is the friction factor which may be correlated as a function of the Reynolds number Re and relative pipe roughness (absolute roughness divided by inside diameter). This function is usually presented in the Moody Diagram:

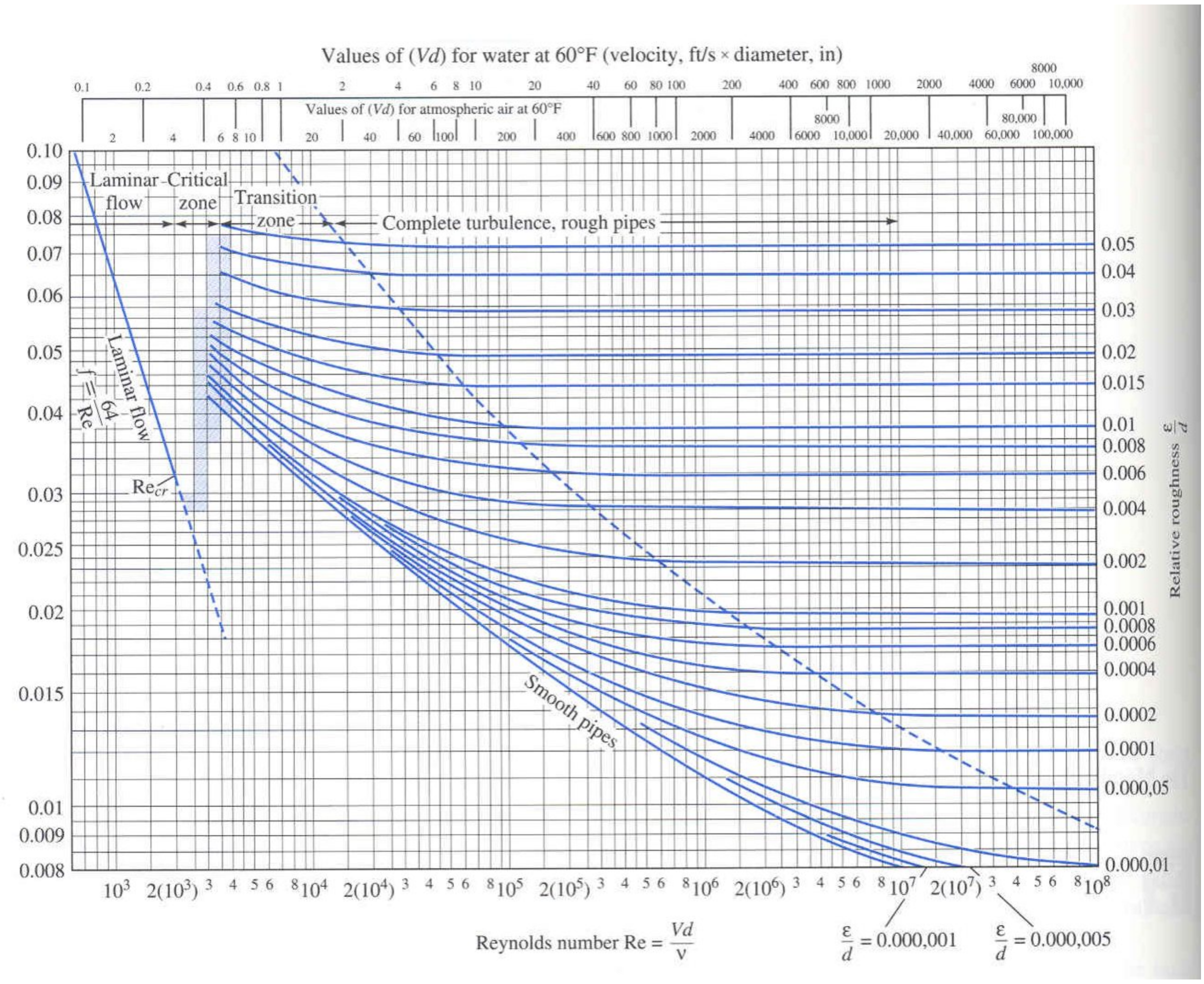


Figure 81: Moody Diagram

Moody diagram may be divided into three zones: laminar, transient and turbulent.

Friction Factor (f)				
Laminar zone	**Hagen-Poiseuille Equation**	$f = \frac{64}{\mathrm{Re}}$	$\mathrm{Re} \le 2000$	
Transient zone	**Blasius Equation**	$f = 0.316\,Re^{-0.25}$	$2000 < \mathrm{Re} \le 5000$	
Turbulent zone	**Colebrook-White Equation (Haaland explicit approximation)**	$f = \frac{1.325}{\left(\ln\left(\frac{\frac{\varepsilon}{D}}{3.7} + \frac{5.74}{\mathrm{Re}^{0.9}}\right)\right)^2}$	$5000 < \mathrm{Re} < 10^8$	$10^{-6} \le \frac{\varepsilon}{D} \le 10^{-2}$

Pipe absolute roughness (ε) may be specified as absolute or variable absolute roughness along pipe in the form of a table. Here absolute roughness is a function of distance from pipe input end (refer to Figure 82).

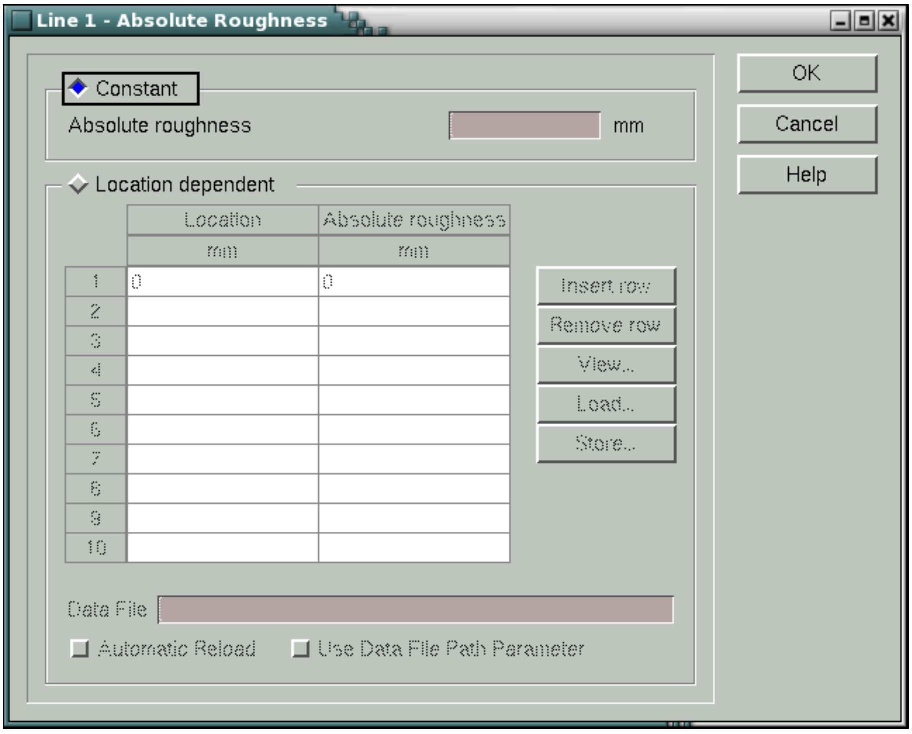


Figure 82: Absolute Roughness Dialog

Bending radius is explained in Section 9.4.6.

Fixed Friction Factor

The wall friction factor may be specified as constant along pipe or as function of distance from the pipe input end (similar to Figure 82):

$$R = -\frac{1}{2}\frac{f(x)}{D}v|v| \quad . \qquad (178)$$

☞ **Note:** Fixed friction factor is typically used for low flow rate with predictable Reynolds number.

Bending radius is explained in Section 9.4.6.

9.4.6. Bending Radius

For the convenience of engineering calculations, the total resistance coefficient of bend is usually determined as the sum of the coefficient of local bend resistance ξ_{loc} and the friction coefficient ξ_{fr}:

$$\xi = \xi_{loc} + \xi_{fr}\,. \tag{179}$$

Coefficient ξ_{fr} is calculated as friction coefficient for straight pipe sections. Coefficient of local bend resistance ξ_{loc} can be obtained from the formulas suggested by Abramovich:

<table>
<tr><td colspan="4" align="center">$\xi_{loc} = A_1 B_1$</td></tr>
<tr><td colspan="2" align="center">$A_1 = f(\beta_B)$</td><td colspan="2" align="center">$B_1 = f\left(\frac{R_B}{D_B}\right)$</td></tr>
<tr><td colspan="2" align="center">A_1 is the coefficient accounting for the effect of the bend and angle β_B.</td><td colspan="2" align="center">B_1 is the coefficient accounting for the relative curvature radius of bend.</td></tr>
<tr><td>$A_1 = 0.9\sin(\beta_B)$</td><td>$\beta_B \le 70^\circ$</td><td rowspan="2">$B_1 = \frac{0.21}{\left(\frac{R_B}{D_B}\right)^{2.5}}$</td><td rowspan="2">$0.5 \le \frac{R_B}{D_B} \le 1.0$</td></tr>
<tr><td>$A_1 = 1.0$</td><td>$\beta_B = 90^\circ$</td></tr>
<tr><td>$A_1 = 0.7 + 0.35\frac{\beta_B}{90^\circ}$</td><td>$\beta_B \ge 100^\circ$</td><td>$B_1 = \frac{0.21}{\sqrt{\frac{R_B}{D_B}}}$</td><td>$\frac{R_B}{D_B} > 1.0$</td></tr>
</table>

In the table D_B is pipe diameter and R_B is bend radius as shown Figure 83.

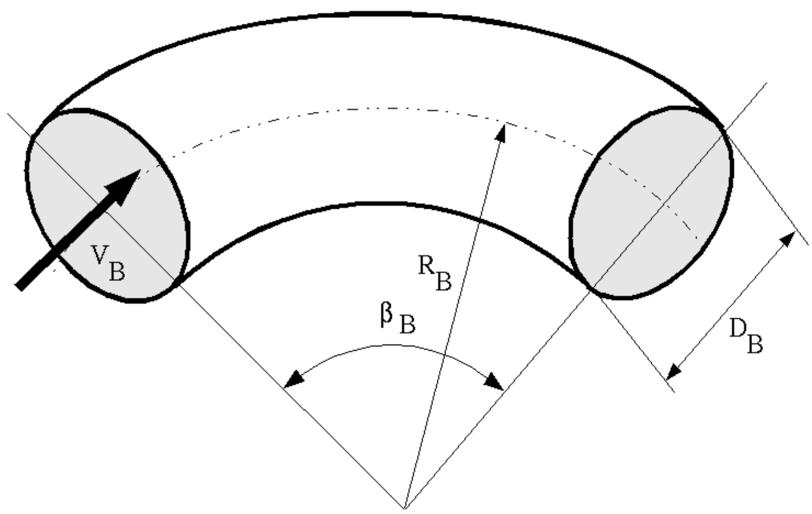


Figure 83: Bend Geometry

The bending radius R_B may be specified as constant along pipe or as function of distance from the pipe input end.

9.4.7. Mitre Bends

Coefficient of local mitre bend resistance ξ_{loc} can be obtained from the formula:

$$\xi_{loc} = k_{\mathrm{Re}} A \xi_{\beta} . \tag{180}$$

Alternatively, resistance coefficient can be found in tables:

$Re \times 10^{-4}$	1.0	1.4	2.0	3.0	4.0	6.0	8.0	10.0	14.0	>20.0
k_{Re}	1.40	1.33	1.26	1.19	1.14	1.09	1.06	1.04	1.0	1.0

β	0.0	20	30	45	60	75	>90
A	-	2.50	2.22	1.87	1.50	1.28	1.20

In the table D_B is pipe diameter and β_B is bend angle as shown Figure 84.

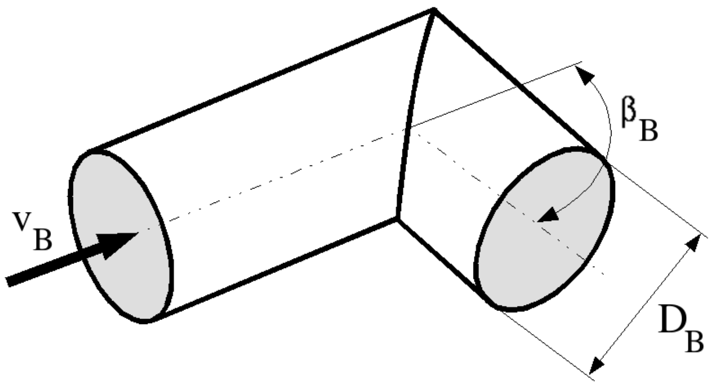


Figure 84: Mitre Bend Geometry

> ☞ **Note:** If line does not contain any mitre bend, then only one row with 0,0 values has to be defined in the table (default).

9.4.8. Equation of Fluid Flow

As stated earlier (refer to Section 9.1.6), one-dimensional isothermal fluid flow in a line is governed by the continuity equation and the equation for the conservation of momentum:

$$\rho_t + (\rho \upsilon)_x = 0 , \tag{181}$$

$$\rho \upsilon_t + \rho \upsilon \upsilon_x + p_x = R , \tag{182}$$

where:

x .... coordinate (in flow direction)

υ .... velocity of fluid in x-direction

p .... pressure

ρ .... fluid density

R .... friction "force" term

The conservation laws (Equations 181 and 182) can be solved only if the velocity $\upsilon(x,t)$ is known a priori or if a relationship between ρ and p is established.

$$p = \mathrm{f}(\rho). \tag{183}$$

This equation is called the equation of state. It depends on physical properties of the fluid under pressure.

System of Equations 181 and 182 can be written in vector form:

$$\mathbf{u_t} + \mathbf{f}(\mathbf{u})_\mathbf{x} = \mathbf{S}\,, \tag{184}$$

where:

$$\mathbf{u} = \begin{pmatrix} \rho \\ \rho\upsilon \end{pmatrix};\ \ \mathbf{f}(\mathbf{u}) = \begin{pmatrix} \rho\upsilon \\ \rho\upsilon^2 + \mathrm{f}(\rho) \end{pmatrix};\ \ \mathbf{S} = \begin{pmatrix} 0 \\ R \end{pmatrix}.$$

Equation 184 can be linearized in the form:

$$\mathbf{u_t} + \mathbf{A}(\mathbf{u})\mathbf{u_x} = \mathbf{S}\,, \tag{185}$$

where $\mathbf{A}(\mathbf{u})$ is a matrix which has to be constructed.

9.4.8.1. CFL Stability Criterion

In the course of the numerical integration of the conservation laws defined by the Equations 181 and 182, special attention has to be focused on the control of the time step. To achieve a stable solution, the CFL criterion (stability criterion defined by Courant, Friedrichs and Lewy) has to be satisfied as described below.

If the number of cells (grid points) is not specified, it is calculated by BOOST Hydsim according to the formula:

$$n_x = \frac{L}{2a\Delta t}\,, \tag{186}$$

where a is velocity of sound (for variable fluid properties initial value or sound velocity is used), Δt is time step and L is line length.

This formula is based on the CFL stability criterion which is checked at each calculation step:

$$\Delta t \leq \frac{\Delta x}{u+a} = \frac{L}{n_x (u+a)}, \tag{187}$$

where Δx is the cell length is given by:

$$\Delta x = \frac{L}{n_x}, \tag{188}$$

and u is the flow velocity in the cell.

Equation 186 is derived by applying the condition $u=a$ (limiting case – sonic flow) in Equation 187.

9.4.9. Godunov Method

Godunov method is a numerical method for solving Equation 185 in conservative form. This implies that a physically relevant solution is certain.

Godunov method is used for solution of the set of non-linear hyperbolic partial differential equations discussed above. It is based on finite volume approach. The solution at the end of time step is obtained from the value at the beginning of time step and from fluxes over the cell borders (refer to Figure 85).

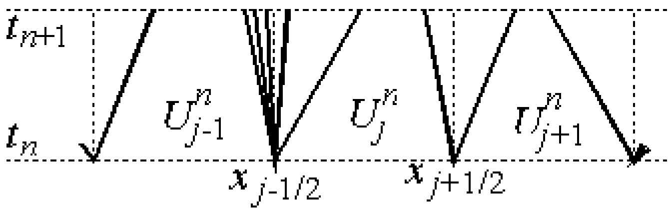


Figure 85: Solution of Independent Riemann Problem

Applying this method, the 3D symmetrical problem is approximated by a 1D problem with independent variables[15]:

$$\int_{x_{j-\frac{1}{2}}}^{x_{j+\frac{1}{2}}} \int_r \int_\varphi \int_{t_k}^{t_{k+1}} u_t r dt d\varphi dr dx + \int_{x_{j-\frac{1}{2}}}^{x_{j+\frac{1}{2}}} \int_r \int_\varphi \int_{t_k}^{t_{k+1}} f(u)_x r dt d\varphi dr dx = \int_v \int_{t_k}^{t_{k+1}} \underbrace{B dt dv}_{R}. \tag{189}$$

The solution of Equation 189 by Godunov method is the following:

$$u_j^{n+1} = u_j^n - \frac{k}{h}\left(f\left(u^*\left(u_j^n, u_{j+1}^n\right)\right) - f\left(u^*\left(u_{j+1}^n, u_j^n\right)\right)\right) - R_j. \tag{190}$$

Inside this equation, the flux at the borders of each cell has to be determined. Normally the flux at the right cell border will not be equal to the flux at the left border of the adjacent cell, which is a necessary condition to meet continuity requirements.

[15] LeVeque, Randall J. *Numerical Methods for Conservation Laws*. Börkhäuser Verlag, Berlin, 1990.

To overcome this problem, a Riemann solver is used to calculate the correct mean value from the two different fluxes at the cell border.

Finite difference discretization of partial differential equations (PDE) is expected to be inappropriate near discontinuities, where the PDEs do not hold. For a first order method as Godunov method, the obtained results are very smeared in regions near the discontinuities. This is because first order numerical methods have a large amount of „numerical viscosity“ that smoothes the solution in much the same way as physical viscosity would. The better discretization of line is, the better are the results of calculation. This phenomenon is illustrated in Figure 86.

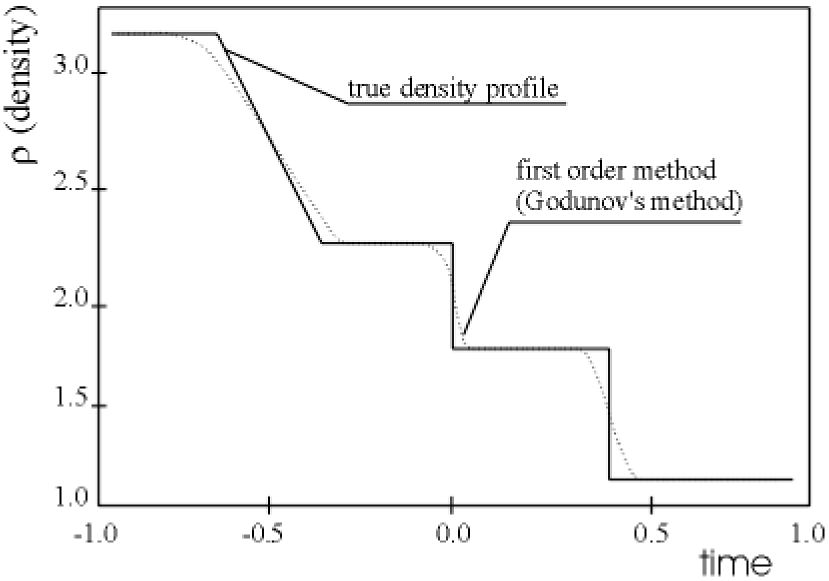


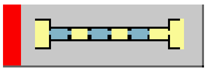
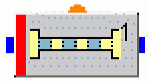
Figure 86: Solution of Shock Tube Problem at t = 0.5

Godunov proposed a method to make use of characteristic information within the framework of a conservative form[16.] Rather than attempting to follow characteristics backwards in time, he suggested solving of Riemann problem forward in time. Solutions of Riemann problem are relatively easy to compute, give substantial information about the characteristic structure, and lead to conservative methods.

Godunov method uses the numerical solution U^n *to* define a piecewise constant function $\tilde{u}^n(x,t_n)$ with the value U_j^n on the grid cell $x_{j-\frac{1}{2}} < x < x_{j+\frac{1}{2}}$. At time *tn* this agrees with the piecewise constant function $U_k(x,t_n)$. However, the function $\tilde{u}^n$, unlike U_k will not be constant over $t_n \le t < t_{n+1}$. Instead, the method uses $\tilde{u}^n(x,t_n)$ as initial data for the conservation law, which is solved exactly to obtain $\tilde{u}^n(x,t)$ for $t_n \le t < t_{n+1}$. The equation can be solved exactly over a short time interval because the initial data $\tilde{u}^n(x,t_n)$ is piecewise constant, and hence defines a sequence of Riemann problems. The exact solution, up to the time when the waves from neighboring Riemann problems begin to interact, is obtained by simply piecing together these Riemann solutions. This is illustrated in Figure 85.

[16] Godunov, S.K. *Matematiceskij Sbornik*, 47, 1959, p. 271 (published in Russian).

9.5. MacCormack Line

Element Name:	**MacCormack Line (single and two-phase)**	
Element Icon:		
Definition:	This element serves to define a line (tube/pipe). The solution of the flow equation is obtained by MacCormack method. Two-phase line contains bubble dynamics (cavitation) model.. Friction losses can be switched off or calculated by various methods.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output connection have to be specified.	

☞

Note: **MacCormack line** model is a 2nd order fluid flow model based on the MacCormack finite difference scheme. This scheme is combined with Flux Corrected Transport (FCT) method. It can be used as a single-phase or two-phase line. For the two-phase flow, the bubble dynamics-cavitation model is applied. **MacCormack line** is the most accurate line model in BOOST Hydsim. It works with strongly variable fluid properties.

Note: Friction in **MacCormack line** can be modeled by **Melcher's non-stationary friction method, Colebrook-White equation** or using the **fixed friction factor MacCormack line** can also be used without friction losses. In all cases it satisfies the mass and momentum conservation laws.

Schematic of **MacCormack Line** with expansions/contractions at the ends is shown in Figure 87.

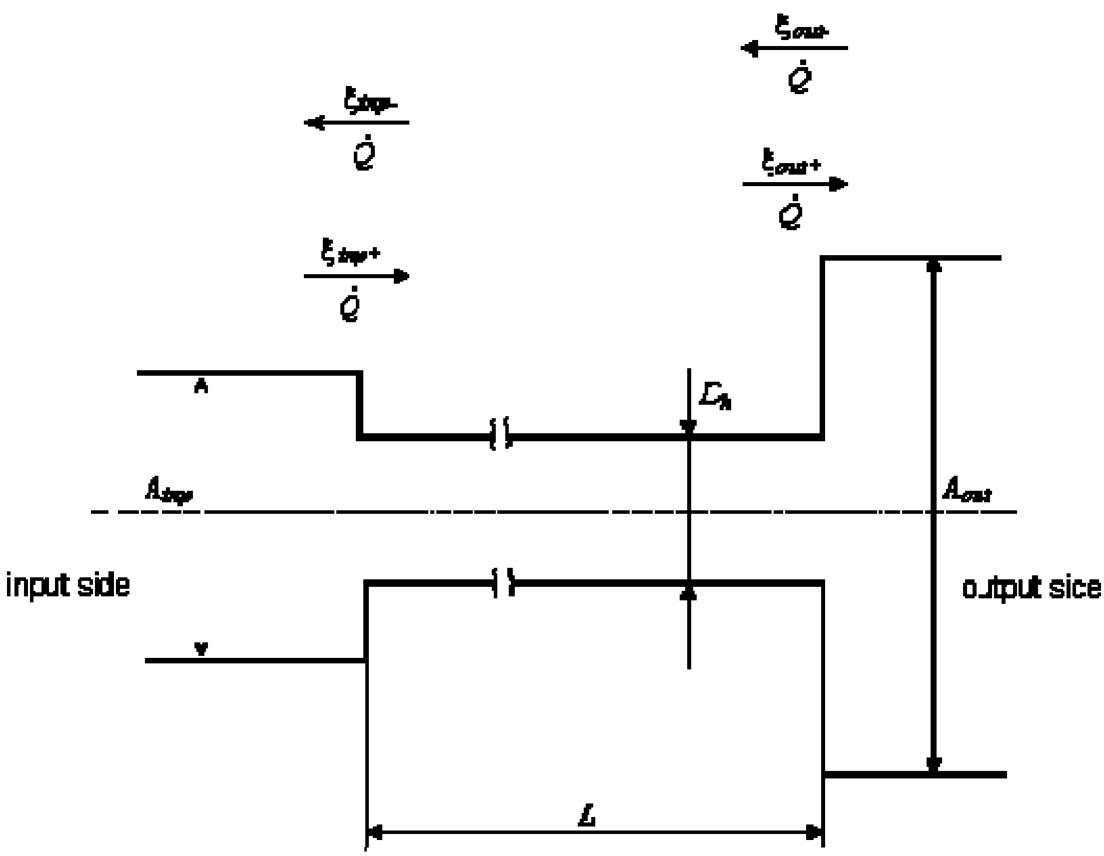


Figure 87: Geometry of MacCormack Line

9.5.1. Input Parameters

Description of input data for the **MacCormack Line**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Friction Losses	Press this bar to specify the model of friction losses to be used in the line. Friction models are explained in Section 9.2.5.
Line length:	Specify length of line. It corresponds to *L* in Figure 79.
Hydraulic diameter:	Specify hydraulic diameter of line. It corresponds to D_h in Figure 79.
Number of line cells:	BOOST Hydsim allows the user to define the number of cells (line divisions). According to it, the time step to cell length ratio is determined at calculation start using the specified initial conditions. Applying Courant-Friedrichs-Lewy (CFL) criterion, this ratio is checked at every time calculation step. CFL criterion is explained in Section 9.4.8.1. If not defined or switched off, number of cells is calculated internally.
Gas Bubble Dynamics	If this button is active, BOOST Hydsim will internally calculate void fraction according to volume pressure.

☞ **Note:** **Gas Bubble Dynamics** is recommended for small gas fraction.

Reference pressure in fluid	Specify reference pressure p_0 at which void fraction and bubble radius are known.
Void fraction at reference pressure	Specify gas fraction β_0 of fluid-gas mixture at reference pressure.
Bubble radius at reference pressure	Specify bubble radius R_0 at reference pressure.
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS:	Refer to Figure 79 for more information.
Cross-sectional area at input end:	Specify cross-sectional area at input end A_{in}. Leaving default value 0 implies no extension/contraction at input end.
Cross-sectional area at output end:	Specify cross-sectional areas at output end A_{out} (refer to Figure 79 for more information). Leaving default value 0 implies no extension/ contraction at input end.
Positive flow resistance coefficient at input end:	Specify resistance coefficient ξ_{in+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Positive flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output ends.
Negative flow resistance coefficient at input end:	Specify resistance coefficients ξ_{in-} for negative flow (from output end to input end) related to A_{nom} at sudden change of cross-sectional area on input end.
Negative flow resistance coefficient at output end:	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to A_{nom} at sudden changes of cross-sectional areas on output ends.

ξ_{in+} and ξ_{in-} have to be defined only if A_{in} is specified (i.e. $A_{inp} \neq A_{nom}$).
ξ_{out+} and ξ_{out-} have to be defined only if A_{out} is specified (i.e. $A_{out} \neq A_{nom}$).

For the estimation of flow resistance coefficients, refer to Section 9.1.5.

Compliant Walls:	This button activates the inclusion of wall compliance into calculation.
Wall thickness:	Enter thickness of line walls *s*.
Young's modulus:	Enter Young's modulus of wall material E_{wall}
Poisson's ratio:	Enter Poisson's ratio of wall material.

The pressure wave propagates along the line with the velocity of sound as given in the equation: $$a = \sqrt{\frac{E}{\rho}}, \quad (191)$$ where: a .... velocity of sound in the fluid ρ .... fluid density E .... resultant elasticity modulus The velocity of sound within the high-pressure lines is also affected by the elasticity of the line walls. This is considered in the equation of the resultant elasticity modulus: $$\frac{1}{E} = \frac{1}{E_{wall}} + \frac{D_h}{s}\frac{1}{E_{fluid}}, \quad (192)$$ where: s .... wall thickness E_{wall} .... Young's modulus of wall material E_{fluid} .... bulk modulus of the fluid The line wall distensions are not considered in MacCormack line model (i.e. D_h = const.).	
Cross-sectional distance from input end:	Define up to 10 line cross-sections by specifying their distance from line input end. For these positions, BOOST Hydsim will calculate pressure and flow rate. This data can be stored on output if activated as output parameters.

9.5.2. Initial Conditions

Initial Conditions of **MacCormack Line** are identical to the **Initial Conditions** of **d'Alembert Line** model and **Laplace Line** model (refer to Section 9.1.2).

9.5.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable Parameters:

hydraulic diameter	SI Unit:	m
wall thickness	SI Unit:	m
cross-sectional area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-sectional area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---

resistance coefficient at output for negative flow	SI Unit:	---
absolute roughness	SI Unit:	m
fixed friction factor	SI Unit:	---
bending radius	SI Unit:	m

9.5.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameter Dialog Box** of **MacCormack Line** is shown in Figure 88.

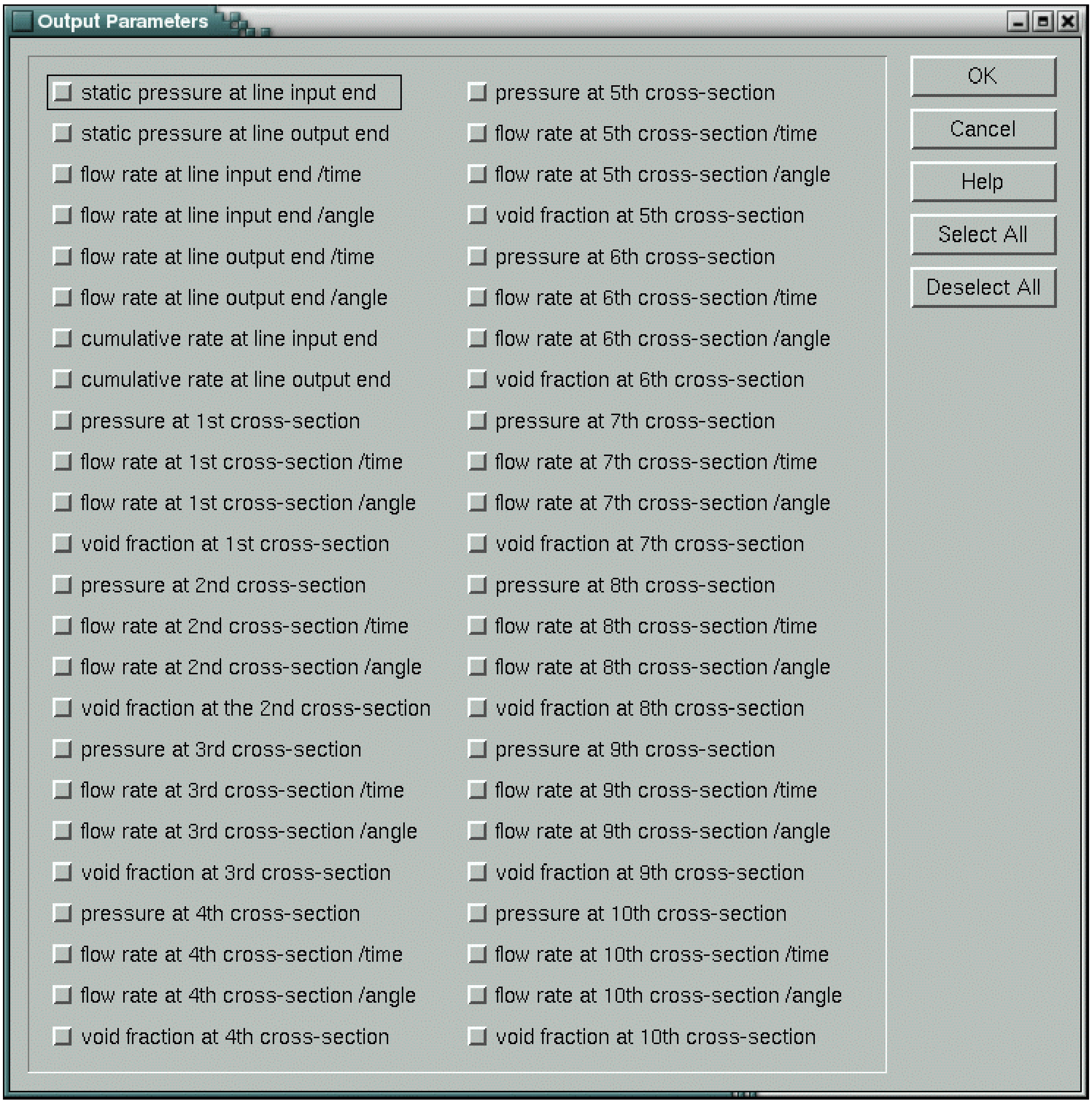


Figure 88: Output Parameters for MacCormack Line Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

9.5.5. MacCormack Explicit Scheme

The MacCormack scheme consists of a predictor and a corrector step. The predictor uses forward finite differences, whereas the backward finite differences are used for the corrector step. Another option for the MacCormack scheme is to use backward finite differences for the predictor step and forward finite differences for the corrector step. A combination of these two options during the computation is recommended by MacCormack in order to achieve a better accuracy. In case of shock occurrence the MacCormack scheme will produce a better resolution of discontinuities when the difference in the predictor is in the direction of propagation of the discontinuity.

The MacCormack scheme for the governing flow equations can be summarized as follows:

Predictor step:

Velocity of the flow

$$\overline{u}_m^{n+1} = u_m^n - \frac{\Delta t}{\Delta x}\left(\left[u_{m+1}^n u_{m+1}^n - u_m^n u_m^n\right] + \left[\frac{p_{m+1}^n}{\rho_{M\,m+1}^{\;n}} - \frac{p_m^n}{\rho_{M\,m}^{\;n}}\right]\right) - \Delta t R_f \, . \tag{193}$$

Pressure in the fluid

$$\overline{p}_m^{n+1} = p_m^n - \frac{\Delta t}{\Delta x}\left(\left[u_{m+1}^n p_{m+1}^n - u_m^n p_m^n\right] + \left[\rho_{M\,m+1}^{\;n} a_{m+1}^{n\;2} u_{m+1}^n - \rho_{M\,m}^{\;n} a_m^{n\;2} u_m^n\right]\right) \tag{194}$$

Corrector step:

Velocity of the flow

$$u_m^{n+1} = \frac{1}{2}\left(u_m^n + \overline{u}_m^{n+1}\right) - \frac{1}{2}\frac{\Delta t}{\Delta x}\left(\left[u_m^n \overline{u}_m^{n+1} - u_{m-1}^n \overline{u}_{m-1}^{n+1}\right] + \left[\frac{\overline{p}_m^{n+1}}{\rho_{M\,m}^{\;n}} - \frac{\overline{p}_{m-1}^{n+1}}{\rho_{M\,m-1}^{\;n}}\right]\right) - \frac{\Delta t}{2}\overline{R}_f \tag{195}$$

Pressure in the fluid

$$p_m^{n+1} = \frac{1}{2}\left(p_m^n + \overline{p}_m^{n+1}\right) - \frac{1}{2}\frac{\Delta t}{\Delta x}\left(\left[u_m^n \overline{p}_m^{n+1} - u_{m-1}^n \overline{p}_{m-1}^{n+1}\right] + \left[\rho_{M\,m}^{\;n} a_m^{n\;2} \overline{u}_m^{n+1} - \rho_{M\,m-1}^{\;n+1} a_{m-1}^{n\;2} \overline{u}_{m-1}^{n+1}\right]\right) \tag{196}$$

The equations above describe the MacCormack scheme where the predictor step is constructed with forward finite differences and the corrector step with backward finite differences. If the MacCormack scheme is applied for the shock problem, the numerical oscillations (‘ripples’) typically occur near the shock profile. To eliminate them, the Flux Corrected Transport method (FCT) is used. This method can damp out numerical oscillations as shown Figure 89 without smearing the shock profile.

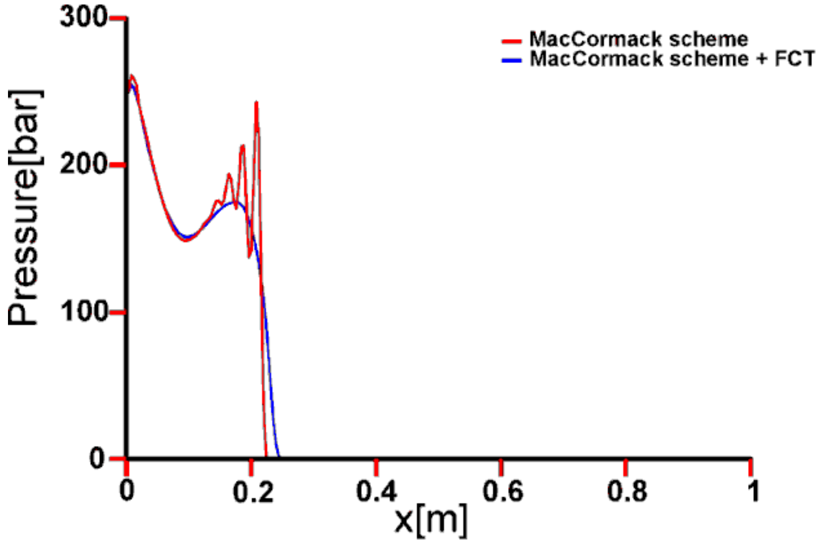


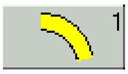
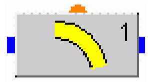
Figure 89: Shock Wave Computed by MacCormack Scheme (with and without FCT)

The FCT algorithm can be rather simply coupled with the MacCormack scheme and is formulated in the following way:

- Introduce a large scale of diffusion in the predictor stage
- Add the anti-diffusion in the corrector stage
- Anti-diffusion stage should be limited to maintain the positive definiteness
- Allow the diffusion to annihilate 'ripples'

10. BEND

10.1. Round/Circular Bend

Element Name:	**Round/Circular Bend**
Element Icon:	
Definition:	This element serves to define the properties of round Bend of circular cross-section and constant area.
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and output hydraulic connection have to be defined. 

> **Note:** Currently the Variable-step solver does not support this element.

Schematic of **Round/Circular Bend** is shown in Figure 90.

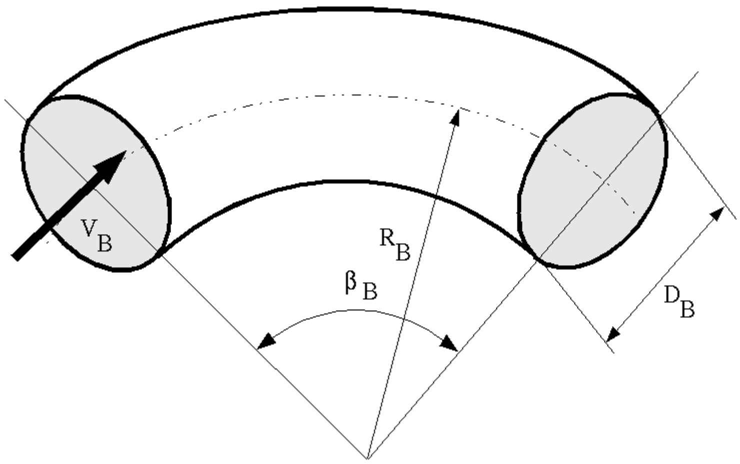


Figure 90: Geometry of Round/Circular Bend

10.1.1. Input Parameters

Description of input data for the **Round/Circular Bend**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Bend diameter:	Specify diameter of bend. It corresponds to D_B in Figure 90.

Bend radius:	Specify radius of bend (curved tube). It corresponds to R_B in Figure 90.
Bend Angle:	Specify angle of bend. It corresponds to β_B in Figure 90.

10.1.2. Initial Conditions

No initial conditions can be specified for **Round/Circular Bend** element.

10.1.3. Modifiable Parameters

Path: **Element** | **Modify**

To activate modification, the button in front of modifiable parameters has to be checked. When modification is activated, the modification table has to be defined. Open it by pressing **Modify...** bar.

Modifiable Parameters:

bend radius:	SI Unit:	m
bend angle:	SI Unit:	rad

10.1.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, activate the check box in front of output parameter.

volumetric flow rate through bend /time	Refer to Section 10.1.6
volumetric flow rate through bend /angle	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow rate through bend	
flow resistance coefficient in bend	Refer to Section 10.1.5

10.1.5. Estimation of Flow Resistance Coefficient

The total resistance coefficient of bend is determined as the sum of the coefficient of local bend resistance ξ_{loc} and the friction coefficient ξ_{fr}

$$\xi = \xi_{loc} + \xi_{fr} \tag{197}$$

Coefficient ξ_{fr} is calculated as friction coefficient for straight section with friction factor f obtained from Hagen-Poiseuille law and from Filonenko-Altshul formula (circular tube with smooth walls):

Friction Factor (*f*)			
Laminar zone	Hagen-Poiseuille Equation	$f = \frac{64}{\mathrm{Re}}$	$\mathrm{Re} \le 2000$

Turbulent zone	Filonenko Equation	$f = \dfrac{1}{(1.8\lg \mathrm{Re} - 1.64)^2}$	$\mathrm{Re} > 2000$

Friction Coefficient (ξ_{fr})
$\xi_{fr} = 0.0175 \dfrac{R_B}{D_B} \beta_B f$

Coefficient of local resistance ξ_{loc} is calculated from the formulas suggested by Abramovich:

<table>
<tr><td colspan="4">$\xi_{loc} = A_1 B_1$</td></tr>
<tr><td colspan="2">$A_1 = f(\beta_B)$</td><td colspan="2">$B_1 = f\left(\dfrac{R_B}{D_B}\right)$</td></tr>
<tr><td colspan="2">A_1 is the coefficient accounting for the effect of the bend and angle β_B.</td><td colspan="2">B_1 is the coefficient accounting for the relative curvature radius of bend.</td></tr>
<tr><td>$A_1 = 0.9\sin(\beta_B)$</td><td>$\beta_B \le 70^\circ$</td><td rowspan="2">$B_1 = \dfrac{0.21}{\left(\dfrac{R_B}{D_B}\right)^{2.5}}$</td><td rowspan="2">$0.5 \le \dfrac{R_B}{D_B} \le 1.0$</td></tr>
<tr><td>$A_1 = 1.0$</td><td>$\beta_B = 90^\circ$</td></tr>
<tr><td>$A_1 = 0.7 + 0.35 \dfrac{\beta_B}{90^\circ}$</td><td>$\beta_B \ge 100^\circ$</td><td>$B_1 = \dfrac{0.21}{\sqrt{\dfrac{R_B}{D_B}}}$</td><td>$\dfrac{R_B}{D_B} > 1.0$</td></tr>
</table>

10.1.6. Flow Equation

Volumetric flow rate through **Round/Circular Bend** is calculated from Bernoulli equation:

$$\dot{Q} = sign(p_{in} - p_{out}) \sqrt{\frac{1}{\xi} \frac{2|p_{in} - p_{out}|}{\rho}}, \tag{198}$$

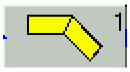
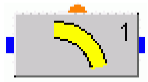
where:

$\dot{Q}$	flow rate
ξ	resistance coefficient
p_{in} and p_{out}	actual pressures on input and output end
ρ	fluid density

Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries).

Note that Equation 198 contains a *sign* function which is necessary to determine the flow direction depending on the pressures on input and output.

10.2. Mitre/Circular Bend

Element Name:	**Mitre/Circular Bend**	
Element Icon:		
Definition:	This element serves to define the properties of mitre Bend of circular cross-section and constant area.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

☞ **Note:** Currently the Variable-step solver does not support this element.

Schematic of **Mitre/Circular Bend** is shown in Figure 91.

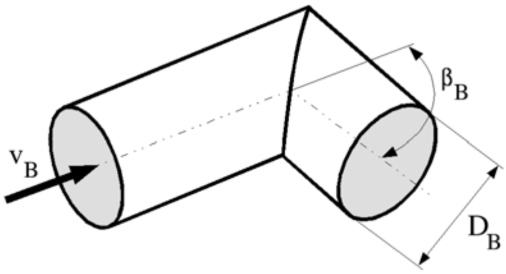


Figure 91: Geometry of Mitre/Circular Bend

10.2.1. Input Parameters

Description of input data for the **Mitre/Circular Bend**:

Element name:	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Bend diameter:	Specify diameter of bend. It corresponds to D_B in Figure 91.
Bend angle:	Specify angle of bend. It corresponds to β_B in Figure 91.

10.2.2. Initial Conditions

Initial conditions cannot be specified for **Mitre/Circular Bend** element.

10.2.3. Modifiable Parameters

Path: **Element** | **Modify**

To activate modification, check the button in front of modifiable parameters. When modification is activated, the modification table has to be defined. Open it by pressing **Modify...** bar.

Modifiable Parameter:

bend angle:	SI Unit:	rad

10.2.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, activate the check box in front of output parameter.

volumetric flow rate through bend /time	Refer to Section 10.2.6
volumetric flow rate through bend /angle	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog box.
flow resistance coefficient in bend	Refer to Section 10.2.5

10.2.5. Estimation of Flow Resistance Coefficient

The total resistance coefficient of bend is determined as the sum of the coefficient of local bend resistance ξ_{loc} and the friction coefficient ξ_{fr} .

$$\xi = \xi_{loc} + \xi_{fr} \quad (199)$$

Coefficient ξ_{fr} is set to zero because it is assumed that there is no bend length.

Coefficient of local resistance ξ_{loc} is calculated from the formula[17]:

$$\xi_{loc} = k_{Re} A \xi_{\beta} , \quad (200)$$

where:

$$\xi_{\beta} = 0.95 \sin^2 \frac{\beta}{2} + 2.05 \sin^4 \frac{\beta}{2} . \quad (201)$$

Coefficient of local resistance can be also obtained from the tables:

$Re \times 10^{-4}$	1.0	1.4	2.0	3.0	4.0	6.0	8.0	10.0	14.0	>20.0
k_{Re}	1.40	1.33	1.26	1.19	1.14	1.09	1.06	1.04	1.0	1.0

[17] Idelchik, I.E., *Handbook of Hydraulic Resistance,* Springer, 2nd Edition, 1986.

β	0.0	20	30	45	60	75	>90
A	-	2.50	2.22	1.87	1.50	1.28	1.20

10.2.6. Flow Equation

Volumetric flow rate through **Mitre/Circular Bend** is calculated from Bernoulli equation:

$$\dot{Q} = sign(p_{in} - p_{out})\sqrt{\frac{1}{\xi}\frac{2|p_{in} - p_{out}|}{\rho}}\,, \tag{202}$$

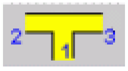
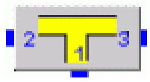
where:

$\dot{Q}$	flow rate
ξ	resistance coefficient
p_{in} and p_{out}	actual pressures on input and output end
ρ	fluid density

Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries).

11. JUNCTION

11.1. Tee (90 deg)

Element Name:	**Tee (90 deg)**	
Element Icon:		
Definition:	This element serves to model Tee Junction with 90 deg angle between Side branch (Port 1) and Straight passage (Port 2 – 3).	
Connecting pins:	standard pins: 3 (hydraulic) special pins: 0 wire pins: 1	

Note: **Junction** element can be connected to two **Lines** and one **Volume** or to three **Line** elements. In the latter case one hydraulic connection must be opposite to the other two, i.e. **Junction** may have either one input and two output connections or one output and two input connections to **Lines**.

Note: **Junction** element takes Port diameters from connected **Line** elements (directly connected or via Volume element). Diameters at Port 2 and 3 must be the same and not smaller than the diameter at Port 1.

11.1.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 7.4.1).
Element connection	Program generated list of the connected elements.
Flow Resistance	Flow resistance coefficient, which is defined as the ratio of the difference of total pressures to the mean velocity pressure in the given section [1]: $\xi = \dfrac{\Delta p}{\rho w^2 / 2}$

[1] I. E. Idelchik, Handbook of Hydraulic Resistance, Second Edition, Springer – Verlag, 1986.

Program Calculated	If this button is active, the internal calculation of flow resistance coefficients will be used.
User Defined	Select to use a user-defined flow Resistance coefficients instead of program-calculated.
Case I	Symmetric Dividing (Diverging) Flow Case
Table 1 – 2	Press this bar to specify flow resistance coefficient from Port 1 to Port 2 in symmetric dividing flow as function of flow rate ratio between flow rate in Port 1 and Port 2. **Note:** Characteristic of Flow resistance coefficient from Port 1 to Port 3 is equal to Flow resistance coefficient from Port 1 to Port 2 due to the same geometry.

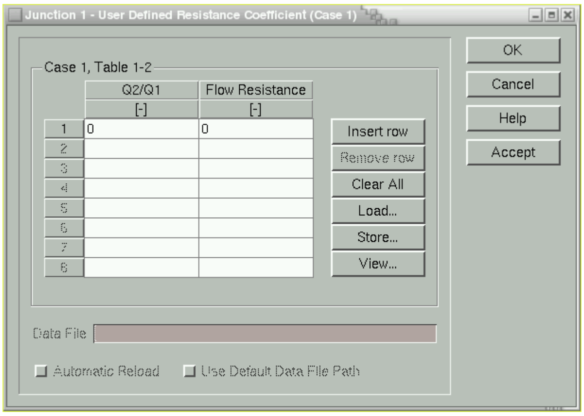


Figure 92: Tee (90 deg) Input Data Dialog

		Specify flow rate ratio between flow rates in Port 1 and Port 2 in ascending order in the first column. Specify flow resistance coefficient from Port 1 to Port 2 in the second column. For flow rate ratios higher than the highest value specified, the highest flow rate ratio value from the table is used. For flow rate ratios lower than the lowest value specified, the lowest flow rate ratio value from the table is taken (kept constant as long as flow rate ratio is out of table range).

Data File	Specify data file from which the data will be loaded. This box becomes active when the Automatic Reload button is active.
Case II	Symmetric Combining (Converging) Flow Case
Table 2 – 1	Press this bar to specify Flow resistance coefficient from Port 2 to Port 1 in symmetric combining flow as function of flow rate ratio between flow rates in Port 2 and Port 1. **Note:** Flow resistance coefficient from Port 3 to Port 1 is equal to the Flow resistance coefficient from Port 2 to Port 1 due to the same geometry. **Note:** Specifications for the Table 2 – 1 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 2 to Port 1.

Case III	Dividing (Diverging) Flow Case
Table 2 – 1	Press this bar to specify Flow resistance coefficient from Port 2 to Port 1 in dividing flow as function of flow rate ratio between flow rates in Port 2 and Port 1. **Note:** Specifications for the Table 2 – 1 are the same as for the Table 1 – 2 (Case **I**), corresponding to the flow rate from Port 2 to Port 1.
Table 2 – 3	Press this bar to specify Flow resistance coefficient from Port 2 to Port 3 in dividing flow as function of flow rate ratio between flow rates in Port 2 and Port 3. **Note:** Specifications for the Table 2 – 3 are the same as for the Table 1 – 2 (Case **I**), corresponding to the flow rate from Port 2 to Port 3.
Case IV	Combining (Converging) Flow Case
Table 1 – 3	Press this bar to specify Flow resistance coefficient from Port 1 to Port 3 in combining flow as function of flow rate ratio between flow rates in Port 1 and Port 3. **Note:** Specifications for the Table 1 – 3 are the same as for the Table 1 – 2 (Case **I**), corresponding to the flow rate from Port 1 to Port 3.
Table 2 – 3	Press this bar to specify Flow resistance coefficient from Port 2 to Port 3 in combining flow as function of flow rate ratio between flow rates in Port 2 and Port 3. **Note:** Specifications for the Table 2 – 3 are the same as for the Table 1 – 2 (Case **I**), corresponding to the flow rate from Port 2 to Port 3.

11.1.2. Initial Conditions

Initial conditions cannot be specified for a **Junction | Tee (90 deg)** element.

11.1.3. Modify Parameter

Modifiable parameters cannot be specified for a **Junction | Tee (90 deg)** element.

11.1.4. Output Parameters

Path: **Element** | **Store Results**

The **Output Parameters Dialog Box** of **Tee Junction (90 deg)** is shown in Figure 93.

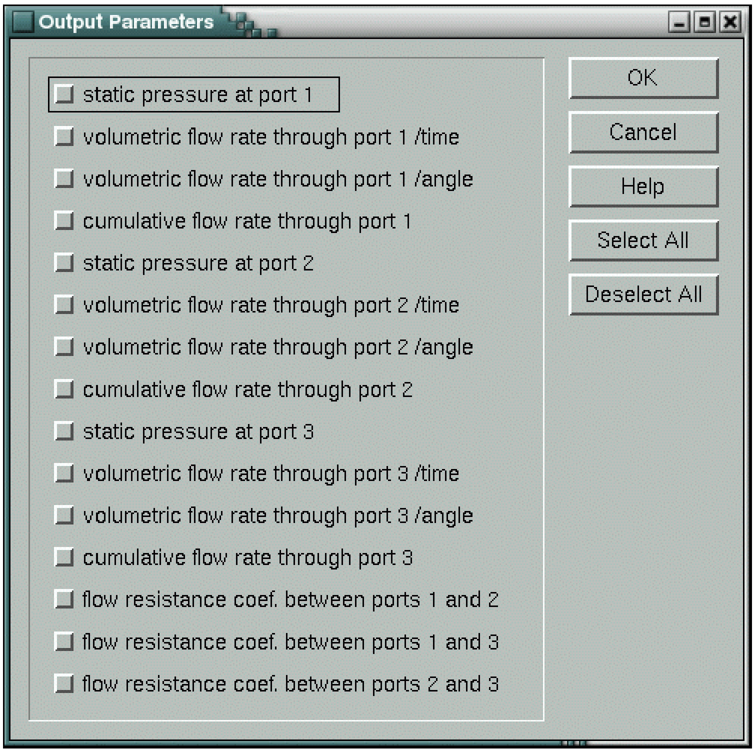


Figure 93: Output Parameters of Tee Junction (90 deg) Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

11.1.5. Tee Junction Model

Tee Junction (90 deg) model is characterized by a straight passage (from Port 2 to Port 3) with the constant diameter ($D_2 = D_3$) and side branch under 90 deg angle from the straight passage. Diameter of the side branch is smaller than or equal to the passage diameter ($D_1 \leq D_2$).

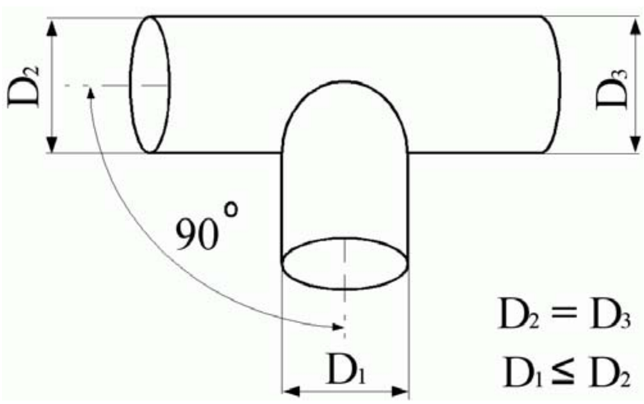


Figure 94: Tee Junction (90 deg) Geometry

11.1.6. Flow Equation

Volumetric flow rate through **Tee Junction** is calculated from the continuity equation:

$$\rho_x Q_x + \rho_y Q_y = \rho_z Q_z \,, \tag{203}$$

where:

Q	flow rate
ρ	flow density
x, y	indices of inflow/ outflow streams
z	index of merged/ divided flow

Two known flow rates are taken from the connected Line elements. The unknown flow rate at the Pressure Port is calculated from equation 203.

Pressure in Pressure Port is taken from the neighboring Volume element (real or virtual) or through inverted connection to the Line element.

11.1.7. Estimation of Flow Resistance Coefficients

The flow resistance coefficient is the ratio of the total pressure loss (drop) to the velocity (dynamic) pressure over an arbitrary flow section. It is determined as the sum of the coefficient of local section resistance ξ_{loc} and the friction coefficient ξ_{fr}.

$$\xi = \xi_{loc} + \xi_{fr} \,, \tag{204}$$

where:

$\xi_{fr} = \dfrac{\Delta p_{fr}}{\rho_{op} w_{op}^2 / 2}$ is the friction loss coefficient in the given flow section of junction

$\xi_{loc} = \dfrac{\Delta p_{loc}}{\rho_{op} w_{op}^2 / 2}$ is the coefficient of local resistance of the given flow section

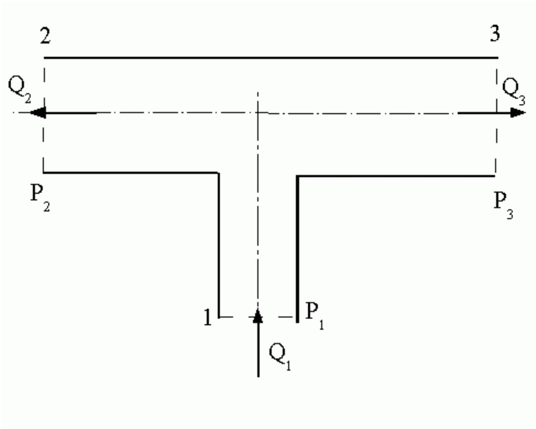
w_{op} is the mean flow velocity at given Port under the operating conditions

ρ_{op} is the fluid density

Flow resistance coefficient is calculated according to the flow type in the Junction. There are four different flow types (cases). For each Flow Case, different equations for the resistance coefficients (two flow sections) are used [19]:

[19] I. E. Idelchik, Handbook of Hydraulic Resistance, Second Edition, Springer – Verlag, 1986.

Case I (Symmetric dividing flow)

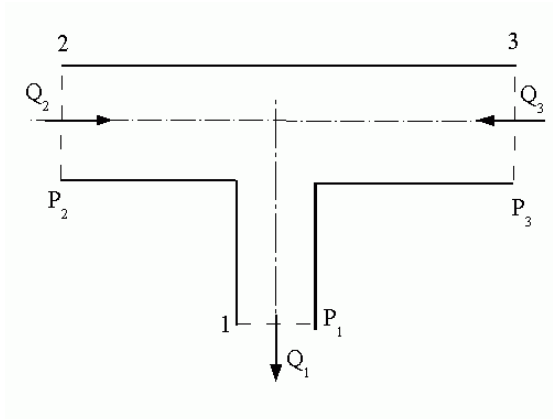


$$\zeta_{1.3} = \frac{\Delta p_3}{\rho_1 w_1^2 / 2} = 1 + \left(\frac{A_1}{A_3}\right)^2 + 3\left(\frac{A_1}{A_3}\right)^2 \left[\left(\frac{Q_3}{Q_1}\right)^2 - \frac{Q_3}{Q_1}\right]$$

$$\zeta_{1.2} = \frac{\Delta p_1}{\rho_1 w_1^2 / 2} = 1 + \left(\frac{A_1}{A_2}\right)^2 + 3\left(\frac{A_1}{A_2}\right)^2 \left[\left(\frac{Q_2}{Q_1}\right)^2 - \frac{Q_2}{Q_1}\right]$$

Δp_1 : pressure drop from Port 2 to Port 1
Δp_3 : pressure drop from Port 3 to Port 1

Case II (Symmetric Combining flow)



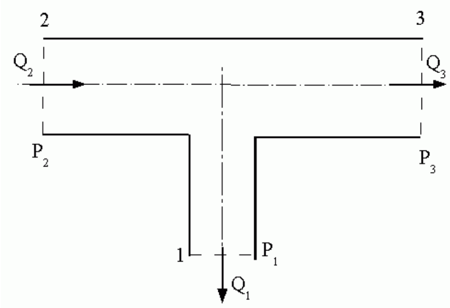
$$\zeta_{1.2} = \frac{\Delta p_1}{\rho_1 w_1^2 / 2} = 1 + k_1 \left(\frac{w_2}{w_1}\right)^2$$

$$\zeta_{1.3} = \frac{\Delta p_3}{\rho_1 w_1^2 / 2} = 1 + k_1 \left(\frac{w_3}{w_1}\right)^2$$

$k_1 = 1.5$

Δp_1 : pressure drop from Port 2 to Port 1
Δp_3 : pressure drop from Port 3 to Port 1

Case III (Dividing flow)



$$\zeta_{2.1} = \frac{\Delta p_1}{\rho_2 w_2^2 / 2} = A \zeta'_{2.1};$$

A: from Table 6 $[A=f(A_1/A_2,\ Q_1/Q_2)]$

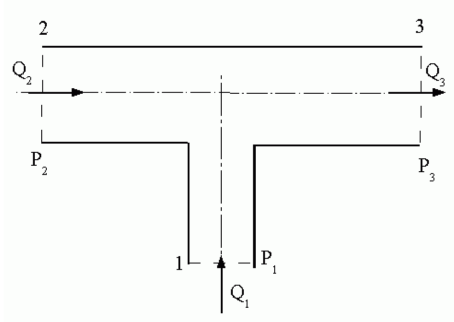
for $\frac{A_1}{A_3} \le \frac{2}{3}$: $\zeta'_{2.1} = 1 + \left(\frac{w_1}{w_2}\right)^2$

for $A_1 = A_3$: $\zeta'_{2.1} = 1 + 0.3\left(\frac{w_1}{w_2}\right)^2$

$$\zeta_{2.3} = \frac{\Delta p_2}{\rho_2 w_2^2 / 2} = 0.4\left(1 - \frac{w_3}{w_2}\right)^2$$

Δp_1 : pressure drop from Port 2 to Port 1
Δp_2 : pressure drop from Port 2 to Port 3

Case IV (Combining flow)



$$\zeta_{3.1} = \frac{\Delta p_3}{\rho_3 w_3^2 / 2} = A\left[1 + \left(\frac{w_1}{w_3}\right)^2 - 2\frac{A_2}{A_3}\left(\frac{w_2}{w_3}\right)^2\right]$$

A: from Table 6 $[A=f(A_1/A_3,\ Q_1/Q_3)]$

$$\zeta_{3.2} = \frac{\Delta p_2}{\rho_3 w_3^2 / 2} = 1.5\frac{Q_1}{Q_3} - \left(\frac{Q_1}{Q_3}\right)^2$$

Δp_3 : pressure drop from Port 1 to Port 3
Δp_2 : pressure drop from Port 2 to Port 3

In the above table the following notations are used:

$\zeta_{x.y}$ ....flow resistance coefficient of flow either from Port Y to Port X (combining flow) or from Port X to Port Y (dividing flow) expressed at merged or divided flow (Port X)

Δp_1 .... absolute pressure drop between Ports 1 and 2

Δp_2 .... absolute pressure drop between Ports 2 and 3

Δp_3 .... absolute pressure drop between Ports 1 and 3

A_1 .... cross-sectional area of Port 1

A_2 .... cross-sectional area of Port 2

A_3 .... cross-sectional area of Port 3

Q_1 .... flow rate at Port 1

Q_2 .... flow rate at Port 2

Q_3 .... flow rate at Port 3

w_1 .... flow velocity at Port 1

w_2 .... flow velocity at Port 2

w_3 .... flow velocity at Port 3

ρ_1 .... fluid density at Port 1

ρ_2 .... fluid density at Port 2

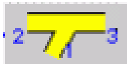
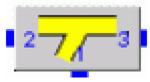
ρ_3 .... fluid density at Port 3

Table 6: Value of Constant *A*

Q_1/Q_3	A_1/A_3	
	≤ 0.35	> 0.35
0– 1.0	*1.0*	
≤ 0.4		$0.9\left(1-\frac{Q_1}{Q_3}\right)$
> 0.4		*0.55*

All pressure drops are calculated relative to the pressure in Pressure Port. This absolute pressure is taken from the connected Volume element (if Volume is connected to Pressure Port) or calculated by inverting connection (if Line element is connected to Pressure Port and **Invert Connections** option is active).

11.2. Tee (angle)

Element Name:	**Tee (angle)**	
Element Icon:		
Definition:	This element serves to model Tee Junction with arbitrary angle between Side branch (Port 1) and Straight passage (Port 2 – 3).	
Connecting pins:	standard pins: 3 (hydraulic) special pins: 0 wire pins: 1	

Note: **Junction** element can be connected to two **Lines** and one **Volume** or to three **Line** elements. In the latter case one hydraulic connection must be opposite to the other two, i.e. **Junction** may have either one input and two output connections or one output and two input connections to **Lines**.

Note: **Junction** element takes Port diameters from connected **Line** elements (directly connected or via Volume element). Diameters at Port 2 and 3 must be the same and not smaller than the diameter at Port 1.

11.2.1. Input Parameters

Description of input data for the **Tee (angle)**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 7.4.1).
Angle Between Ports 1 and 2	Specify angle between side branch (Port 1) and Port 2 (refer to Figure 96)
Element connection	Program generated list of the connected elements.
Flow Resistance	Flow resistance coefficient, which is defined as the ratio of the difference of total pressures to the mean velocity pressure in the given section [20]: $$\xi = \frac{\Delta p}{\rho w^2 / 2}$$
Program Calculated	If this button is active, the internal calculation of flow resistance coefficients will be used.

[20] I. E. Idelchik, Handbook of Hydraulic Resistance, Second Edition, Springer – Verlag, 1986.

User Defined	Check to use a user-defined flow resistance coefficients instead of program-calculated.
Case I	Dividing (Diverging) Flow Case
Table 1 – 2	Press this bar to specify Flow resistance coefficient from Port 1 to Port 2 in dividing flow as function of flow rate ratio between flow rate in Port 1 and Port 2. Specify flow rate ratio between flow rates in Port 1 and Port 2 in ascending order in the first column. Specify flow resistance coefficient from Port 1 to Port 2 in the second column. For flow rate ratios higher than the highest value specified, the highest flow rate ratio value from the table is used. For flow rate ratios lower than the lowest value specified, the lowest flow rate ratio value from the table is taken (kept constant as long as flow rate ratio is out of table range).
Data File	Specify data file from which the data will be loaded. This box becomes active when the Automatic Reload button is active.
Table 1 – 3	Press this bar to specify Flow resistance coefficient from Port 1 to Port 3 in dividing flow as function of flow rate ratio between flow rates in Port 1 and Port 3. **Note:** Specifications for the Table 1 – 3 are the same as for the Table 1 – 2, corresponding to the flow rate from Port 1 to Port 3.
Case II	Combining (Converging) Flow Case
Table 2 – 1	Press this bar to specify Flow resistance coefficient from Port 2 to Port 1 in combining flow as function of flow rate ratio between flow rates in Port 2 and Port 1. **Note:** Specifications for the Table 2 – 1 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 2 to Port 1.
Table 3 – 1	Press this bar to specify Flow resistance coefficient from Port 3 to Port 1 in combining flow as function of flow rate ratio between flow rates in Port 3 and Port 1. **Note:** Specifications for the Table 3 – 1 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 3 to Port 1.
Case III	Backward Dividing (Diverging) Flow Case. **Note:** If angle between Ports 1 and 2 is greater than 90 deg, flow type is forward dividing flow.
Table 2 – 1	Press this bar to specify Flow resistance coefficient from Port 2 to Port 1 in backward dividing flow as function of flow rate ratio between flow rates in Port 2 and Port 1. **Note:** Specifications for the Table 2 – 1 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 2 to Port 1.

Table 2 – 3	Press this bar to specify Flow resistance coefficient from Port 2 to Port 3 in backward dividing flow as function of flow rate ratio between flow rates in Port 2 and Port 3. **Note:** Specifications for the Table 2 – 3 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 2 to Port 3.
Case IV	Forward Dividing (Diverging) Flow Case. **Note:** If angle between Ports 1 and 2 is greater than 90 deg, flow type is backward dividing flow.
Table 3 – 1	Press this bar to specify Flow resistance coefficient from Port 3 to Port 1 in forward dividing flow as function of flow rate ratio between flow rates in Port 3 and Port 1. **Note:** Specifications for the Table 3 – 1 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 3 to Port 1.
Table 3 – 2	Press this bar to specify Flow resistance coefficient from Port 3 to Port 1 in forward dividing flow as function of flow rate ratio between flow rates in Port 2 and Port 3. **Note:** Specifications for the Table 3 – 2 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 3 to Port 2.
Case V	Forward Combining (Converging) Flow Case. **Note:** If angle between Ports 1 and 2 is greater than 90 deg, flow type is backward combining flow.
Table 1 – 3	Press this bar to specify Flow resistance coefficient from Port 1 to Port 3 in forward combining flow as function of flow rate ratio between flow rates in Port 1 and Port 3. **Note:** Specifications for the Table 1 – 3 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 1 to Port 3.
Table 2 – 3	Press this bar to specify Flow resistance coefficient from Port 2 to Port 3 in forward combining flow as function of flow rate ratio between flow rates in Port 2 and Port 3. **Note:** Specifications for the Table 2 – 3 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 2 to Port 3.
Case VI	Backward Combining (Converging) Flow Case. **Note:** If angle between Ports 1 and 2 is greater than 90 deg, flow type is forward combining flow.
Table 1 – 2	Press this bar to specify Flow resistance coefficient from Port 1 to Port 2 in backward combining flow as function of flow rate ratio between flow rates in Port 1 and Port 2. **Note:** Specifications for the Table 1 – 2 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 1 to Port 2.

Table 3 – 2	Press this bar to specify Flow resistance coefficient from Port 3 to Port 2 in backward combining flow as function of flow rate ratio between flow rates in Port 3 and Port 2. **Note:** Specifications for the Table 3 – 2 are the same as for the Table 1 – 2 (Case I), corresponding to the flow rate from Port 3 to Port 2.

11.2.2. Initial Conditions

Initial conditions cannot be specified for a **Junction | Tee (angle)** element.

11.2.3. Modify Parameter

Modifiable parameter cannot be specified for a **Junction | Tee (angle)** element.

11.2.4. Output Parameters

Path: **Element | Store Results**

The **Output Parameters Dialog Box** of **Tee Junction (angle)** is shown in Figure 95.

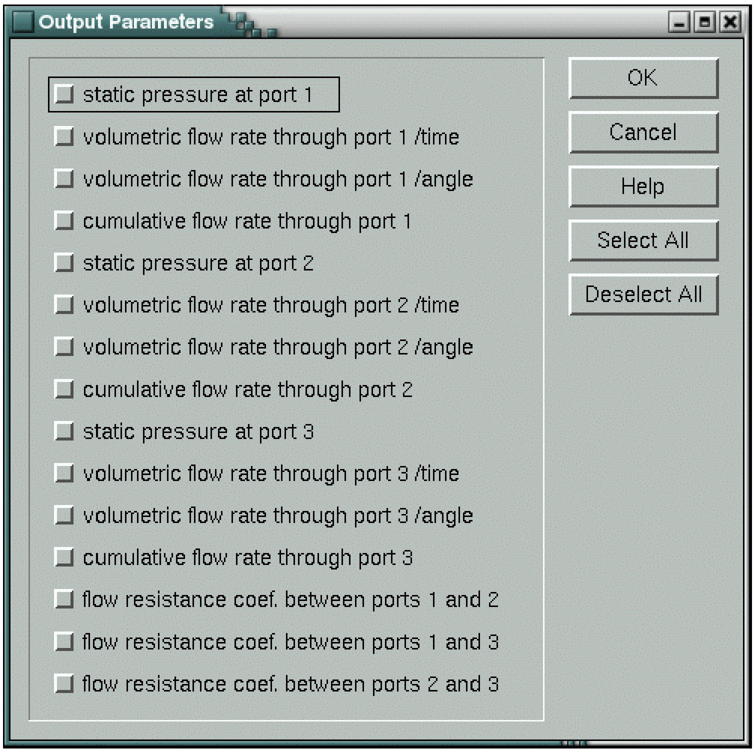


Figure 95: Output Parameters of Tee Junction (angle) Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

11.2.5. Tee Junction Model

The **Tee Junction (angle)** model is characterized by the straight passage (from Port 2 to Port 3) with constant diameter ($D_2 = D_3$) and angle from 0 to 180 deg between the straight passage (Port 2 – 3) and side branch (Port 1). Diameter of the side branch is smaller than or equal to the passage diameter ($D_1 \leq D_2$).

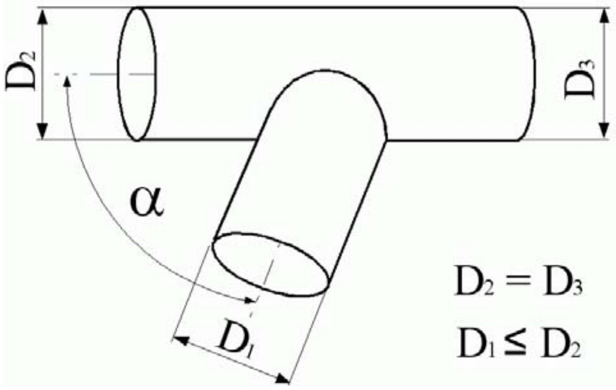


Figure 96: Tee Junction (angle) Geometry

11.2.6. Flow Equation

Volumetric flow rate through **Tee Junction** is calculated from continuity equation:

$$\rho_x Q_x + \rho_y Q_y = \rho_z Q_z \,, \tag{205}$$

where:

Q	flow rate
ρ	flow density
x, y	indexes of inflow/ outflow streams
z	index of merged/ divided flow

Two known flow rates are taken from the connected Line elements and unknown flow rate at the pressure Port is calculated from equation 203.

Pressure in Pressure Port is taken from the neighboring Volume element (real or virtual) or through inverted connection to the Line element.

11.2.7. Estimation of Flow Resistance Coefficients

The Flow Resistance coefficient is the ratio of the total pressure loss (drop) to the velocity (dynamic) pressure over an arbitrary flow section. It is determined as the sum of the coefficient of local section resistance ξ_{loc} and the friction coefficient ξ_{fr}.

$$\xi = \xi_{loc} + \xi_{fr} \,, \tag{206}$$

where:

$\xi_{fr} = \dfrac{\Delta p_{fr}}{\rho_{op} w_{op}^2 / 2}$ is the friction loss coefficient in the given flow section of junction

$\xi_{loc} = \dfrac{\Delta p_{loc}}{\rho_{op} w_{op}^2 / 2}$ is the coefficient of local resistance of the given flow section

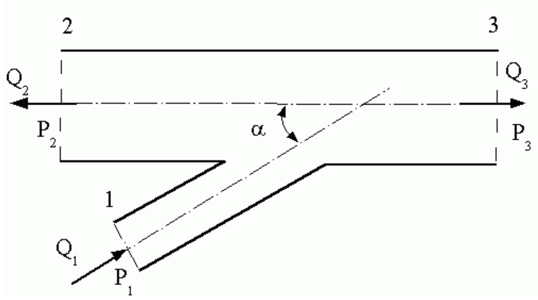
w_{op} is the mean flow velocity at given Port under the operating conditions

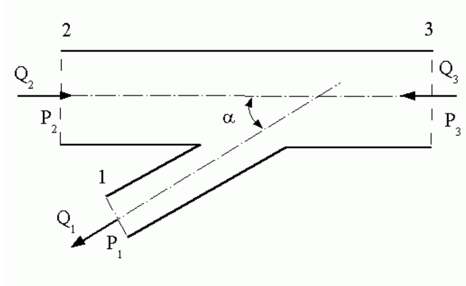
ρ_{op} is the density

Flow resistance coefficient is calculated according to the Flow type in the Junction. There are six different Flow types (Cases) and for each Flow Case, different equations for both resistance coefficients (two flow sections) are used.

Furthermore, according to the angle between Ports 1 and 2 there are three different groups of flow cases:

- Junction with *angle < 90 deg*
- Junction with *angle = 90 deg*
- Junction with *angle > 90 deg*

<table>
<tr><th colspan="2">ANGLE < 90 deg</th></tr>
<tr><th>CaseI (Dividing flow)</th><th>Case II (Combining flow)</th></tr>
<tr><td>


Diagrams of dividing flow for (no interpolation):
- $\alpha = 30°, 45°$ and $60°$
- (A_1/A_3) = 0.5 and 1
$\zeta_{1.2} = \dfrac{\Delta p_1}{\rho_1 w_1^2 / 2}$
$\zeta_{1.3} = \dfrac{\Delta p_3}{\rho_1 w_1^2 / 2}$
Δp_1 : pressure drop from Port 1 to Port 2
Δp_3 : pressure drop from Port 1 to Port 3
</td><td>


Diagrams of combining flow for (no interpolation):
- $\alpha = 30°, 45°$ and $60°$
- (A_1/A_3) = 0.5 and 1
$\zeta_{1.2} = \dfrac{\Delta p_1}{\rho_1 w_1^2 / 2}$
$\zeta_{1.3} = \dfrac{\Delta p_3}{\rho_1 w_1^2 / 2}$
Δp_1 : pressure drop from Port 1 to Port 2
Δp_3 : pressure drop from Port 1 to Port 3
</td></tr>
</table>

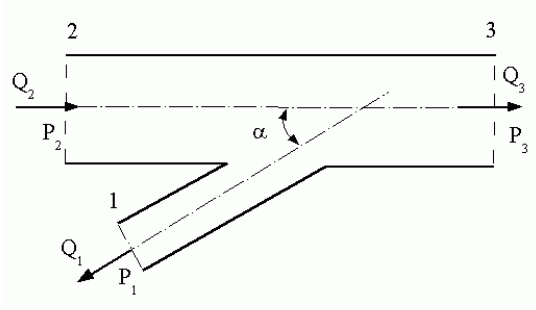
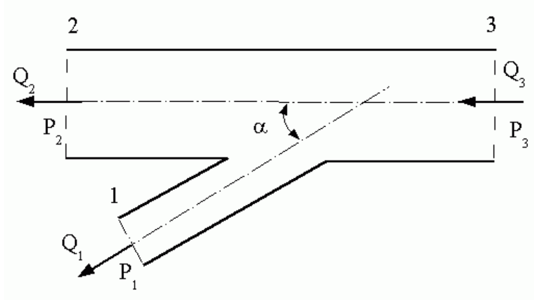
<table>
<tr><th>Case III (Backward dividing flow)[21]</th><th>Case IV (Forward dividing flow)[22]</th></tr>
<tr><td>


Fig 13.22 ⇒ $K_{21}=f(A_1/A_2,Q_1/Q_2)$ for α = 60° (interpolated)

$$K_{21} = \frac{\Delta p_1}{\rho_2 w_2^2 / 2}$$

Fig 13.23 ⇒ $K_{23}=f(Q_1/Q_2)$ for α = 45°-90° (interpolated)

$$K_{23} = \frac{\Delta p_2}{\rho_2 w_2^2 / 2}$$

Δp_1 : pressure drop from Port 2 to Port 1
Δp_2 : pressure drop from Port 2 to Port 3
</td><td>


$$\zeta_{3.1} = \frac{\Delta p_3}{\rho_3 w_3^2 / 2} = A\left[1+\left(\frac{w_1}{w_3}\right)^2 - 2\frac{w_1}{w_3}\cos\alpha\right]$$

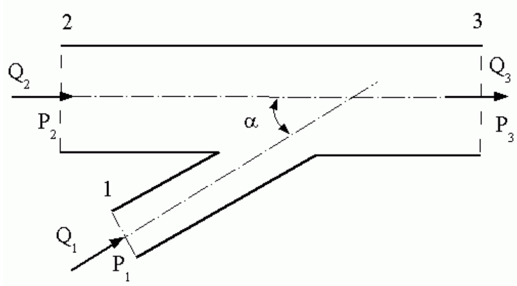
A: from **Table 6** *[A=f(A1/A3, Q1/Q3)]*

$$\zeta_{3.2} = \frac{\Delta p_2}{\rho_3 w_3^2 / 2} = 0.4\left(1-\frac{w_2}{w_3}\right)^2$$

Δp_2 : pressure drop from Port 3 to Port 2
Δp_3 : pressure drop from Port 3 to Port 1
</td></tr>
</table>

[21] D.S.Miller: Internal Flow System, 2nd Edition, BHRA, 1986.
[22] I. E. Idelchik, Handbook of Hydraulic Resistance, Second Edition, Springer – Verlag, 1986.

Case V (Forward combining flow) [5]



$$\zeta_{3.1} = \frac{\Delta p_3}{\rho_3 w_3^2/2} = A\left[1+\left(\frac{w_1}{w_3}\right)^2 - 2\frac{A_2}{A_3}\left(\frac{w_2}{w_3}\right)^2 - 2\frac{A_1}{A_3}\left(\frac{w_1}{w_3}\right)^2 cos\,\alpha\right]$$

A: from **Table 6** *[A=f(A*$_1$*/A*$_3$*, Q*$_1$*/Q*$_3$*)]*

for $\alpha \leq 60°$**:**

$$\zeta_{3.2} = \frac{\Delta p_2}{\rho_3 w_3^2/2} = 1+\left(\frac{w_2}{w_3}\right)^2 - 2\frac{A_2}{A_3}\left(\frac{w_2}{w_3}\right)^2 - 2\frac{A_1}{A_3}\left(\frac{w_1}{w_3}\right)^2 \cos\alpha$$

for $\alpha > 60°$**:**

$$\zeta_{3.2} = \frac{\Delta p_2}{\rho_3 w_3^2/2} = 1.5\frac{Q_1}{Q_3} - \left(\frac{Q_1}{Q_3}\right)^2$$

Δp_3: pressure drop from Port 1 to Port 3

Δp_2: pressure drop from Port 2 to Port 3

Case VI (Backward combining flow) [4]

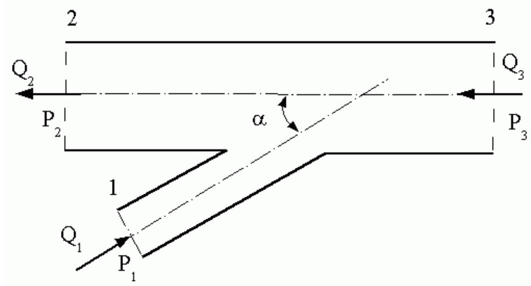


Fig 13.12 $\Rightarrow K_{12} = f(A_1/A_2, Q_1/Q_2)$ for α = 60° (interpolated)

$$K_{12} = \frac{\Delta p_1}{\rho_2 w_2^2/2}$$

Fig 13.13 $\Rightarrow K_{32} = f(A_1/A_2, Q_1/Q_2)$ for α = 60° (interpolated)

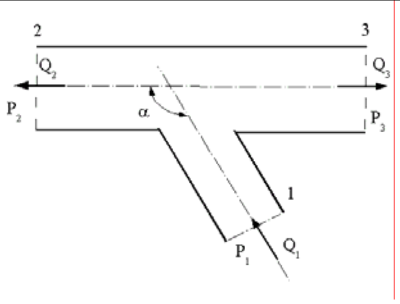
$$K_{32} = \frac{\Delta p_2}{\rho_2 w_2^2/2}$$

Δp_1: pressure drop from Port 1 to Port 2

Δp_2: pressure drop from Port 3 to Port 2

ANGLE = 90 deg

CaseI (Symmetric Dividing flow)	***Case II (Symmetric Combining flow)***
refer to *Section **11.1.7** – Case I*	refer to *Section **11.1.7** – Case II*
Case III (Dividing flow)	***Case IV (Dividing flow)***
refer to *Section **11.1.7** – Case III*	refer to *Section **11.1.7** – Case III*
Case V (Combining flow)	***Case VI (Combining flow)***
refer to *Section **11.1.7** – Case IV*	refer to *Section **11.1.7** – Case IV*

<table>
<tr><th colspan="2">ANGLE > 90 deg</th></tr>
<tr><th>CaseI (Dividing flow)</th><th>Case II (Combining flow)</th></tr>
<tr><td>


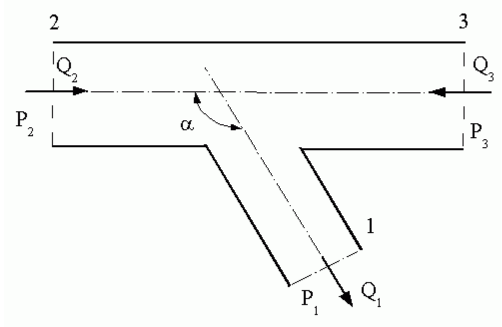
Diagrams of dividing flow for (no interpolation):

- β(=180°- α) = 30°, 45° and 60°
- (A_1/A_3) = 0.5 and 1

$$\zeta'_{1.2} = \frac{\Delta p_1}{\rho_1 w_1^2/2} = \zeta_{1.3}(for \quad \alpha < 90°)$$

$$\zeta'_{1.3} = \frac{\Delta p_3}{\rho_1 w_1^2/2} = \zeta_{1.2}(for \quad \alpha < 90°)$$

Δp_1: pressure drop from Port 1 to Port 2

Δp_3: pressure drop from Port 1 to Port 3
</td><td>


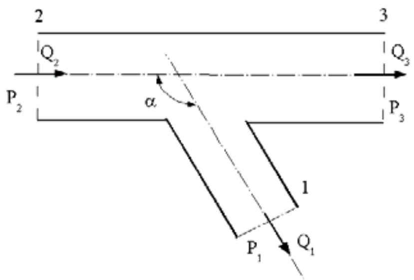
Diagrams of dividing flow for (no interpolation):

- β(=180°- α) = 30°, 45° and 60°
- (A_1/A_3) = 0.5 and 1

$$\zeta'_{1.2} = \frac{\Delta p_1}{\rho_1 w_1^2/2} = \zeta_{1.3}(for \quad \alpha < 90°)$$

$$\zeta'_{1.3} = \frac{\Delta p_3}{\rho_1 w_1^2/2} = \zeta_{1.2}(for \quad \alpha < 90°)$$

Δp_1: pressure drop from Port 2 to Port 1

Δp_3: pressure drop from Port 3 to Port 1
</td></tr>
<tr><th>Case III (Backward dividing flow)[23]</th><th>Case IV (Forward dividing flow)[24]</th></tr>
<tr><td>


$$\zeta_{2.1} = \frac{\Delta p_1}{\rho_2 w_2^2/2} = A'\left[\begin{array}{l} 1+\left(\frac{w_1}{w_2}\right)^2 \\ -2\frac{w_1}{w_2}\cos\beta \end{array}\right]$$

$\beta = 180° - \alpha$; $\beta \leq 60°$; A': from **Table 6**

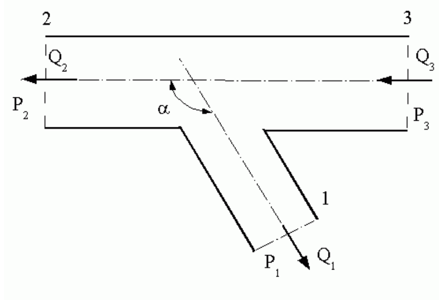
$$\zeta_{2.3} = \frac{\Delta p_2}{\rho_2 w_2^2/2} = 0.4\left(1-\frac{w_3}{w_2}\right)^2$$
</td><td>


Fig 13.22 ⇒ $K_{31}=f(A_1/A_3, Q_1/Q_3)$ for α = 120° (interpolated)

$$K_{31} = \frac{\Delta p_3}{\rho_3 w_3^2/2}$$

Fig 13.23 ⇒ $K_{32}=f(Q_1/Q_3)$ for α = 45°-90° (interpolated)

$$K_{32} = \frac{\Delta p_2}{\rho_3 w_3^2/2}$$
</td></tr>
</table>

[23] I. E. Idelchik, Handbook of Hydraulic Resistance, Second Edition, Springer – Verlag, 1986.
[24] D.S.Miller: Internal Flow System, 2nd Edition, BHRA, 1986.

Δp_1 : pressure drop from Port 2 to Port 1 Δp_2 : pressure drop from Port 2 to Port 3	Δp_3 : pressure drop from Port 3 to Port 1 Δp_2 : pressure drop from Port 3 to Port 2
Case V (Forward combining flow) [7]	***Case VI (Backward combining flow)*** [6]

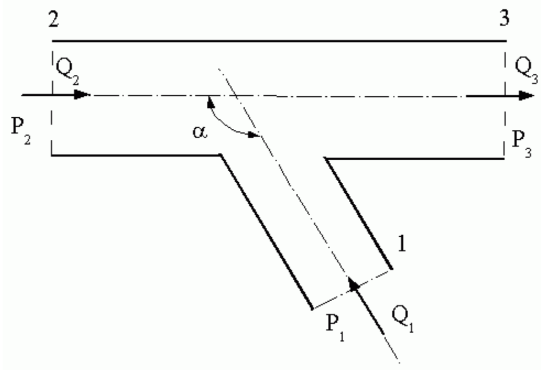
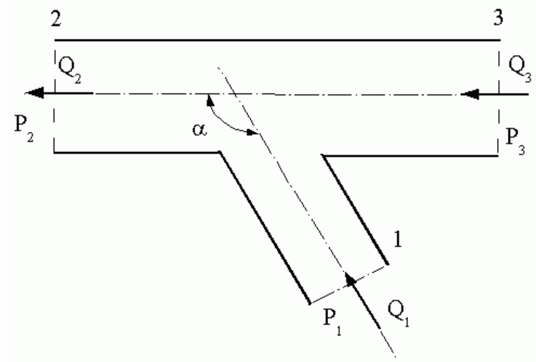

	

Fig 13.12 $\Rightarrow K_{13} = f(A_1/A_3, Q_1/Q_3)$ for α = 120° (interpolated) $$K_{13} = \frac{\Delta p_3}{\rho_3 w_3^2 / 2}$$ Fig 13.13 $\Rightarrow K_{23} = f(A_1/A_3, Q_1/Q_3)$ for α = 120° (interpolated) $$K_{23} = \frac{\Delta p_2}{\rho_3 w_3^2 / 2}$$ Δp_3 : pressure drop from Port 1 to Port 3 Δp_2 : pressure drop from Port 2 to Port 3	$$\zeta_{2.1} = \frac{\Delta p_1}{\rho_2 w_2^2 / 2} = A\left[1+\left(\frac{w_1}{w_2}\right)^2 - 2\frac{A_3}{A_2}\left(\frac{w_3}{w_2}\right)^2 - 2\frac{A_1}{A_2}\left(\frac{w_1}{w_2}\right)^2 \cos\beta\right]$$ A: from **Table 6** $\beta = 180° - \alpha \,;\, \beta \le 60°$: $$\zeta_{2.3} = \frac{\Delta p_2}{\rho_2 w_2^2 / 2} = 1+\left(\frac{w_3}{w_2}\right)^2 - 2\frac{A_3}{A_2}\left(\frac{w_3}{w_2}\right)^2 - 2\frac{A_1}{A_2}\left(\frac{w_1}{w_2}\right)^2 \cos\beta$$ $\beta > 60°$: $$\zeta_{3.2} = \frac{\Delta p_2}{\rho_2 w_2^2 / 2} = 1.5\frac{Q_1}{Q_2} - \left(\frac{Q_1}{Q_2}\right)^2$$ Δp_1 : pressure drop from Port 1 to Port 2 Δp_2 : pressure drop from Port 3 to Port 2

In the above table the following notations are used:

$\zeta_{x.y}$ ....flow resistance coefficient of flow either from Port Y to Port X (combining flow) or from Port X to Port Y (dividing flow) expressed at merged or divided flow (Port X)

Δp_1 .... absolute pressure drop between Ports 1 and 2

Δp_2 .... absolute pressure drop between Ports 2 and 3

Δp_3 .... absolute pressure drop between Ports 1 and 3

A_1 .... cross-sectional area of Port 1

A_2 .... cross-sectional area of Port 2

A_3 .... cross-sectional area of Port 3

Q_1 .... flow rate at Port 1

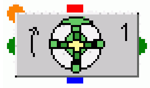
Q_2 .... flow rate at Port 2

Q_3 .... flow rate at Port 3
w_1 .... flow velocity at Port 1
w_2 .... flow velocity at Port 2
w_3 .... flow velocity at Port 3
ρ_1 .... fluid density at Port 1
ρ_2 .... fluid density at Port 2
ρ_3 .... fluid density at Port 3

All pressure drops are calculated relative to the pressure in Pressure Port. This absolute pressure is taken from the connected Volume element (if Volume is connected to Pressure Port) or calculated by inverting connection (if Line element is connected to Pressure Port and **Invert Connections** option is active).

12. PUMP

12.1. Radial Piston Distributor Pump

Element Name:	**Radial Piston Distributor (RPD) Pump**	
Element Icon:		
Definition:	This element serves to define a Radial Piston Distributor (RPD) pump. The model can account for elastic Hertz contact between the pump parts.	
Connecting pins:	standard pins: 2 (1 mechanical and 1 hydraulic) special pins: 2 wire pins: 1	

☞ **Note:** Currently the Variable-step solver does not support this element.

☞ **Note:** Mechanical connection can be defined only on input end and only in w-direction (rotation of the rotor). Hydraulic connection must be defined on output end (x-direction).

12.1.1. Operation Principle

RPD Pump is a rotary diesel injection pump. Contrary to the conventional injection pumps, the rotating part of the RPD Pump there is not the cam profile (camshaft), but the interior parts of the pump body (rotor). Figure 97 shows the schematic front view of a RPD pump with four pistons.

The pump contains a ring with the cam contour on the inner side, called the scrolring shown in Figure 97. The rotor has one or two full-length radial bores in which the pistons are moving. Each piston contains a shoe with a cylindrical roller. Dynamic model of scrolring-roller and shoe-piston connection are shown in Figure 98.

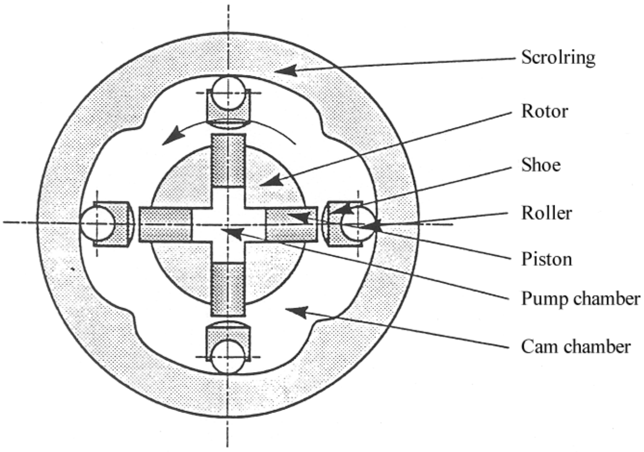


Figure 97: Schematic of RPD Pump (front view)

> ☞ **Note:** Bending of the rotor is impossible due to the symmetrical location of the pistons. **RPD Pump** element does not include the pump chamber (refer to Figure 97). Chamber must be modeled as a separate **Volume** element connected to Pump by output hydraulic connection.

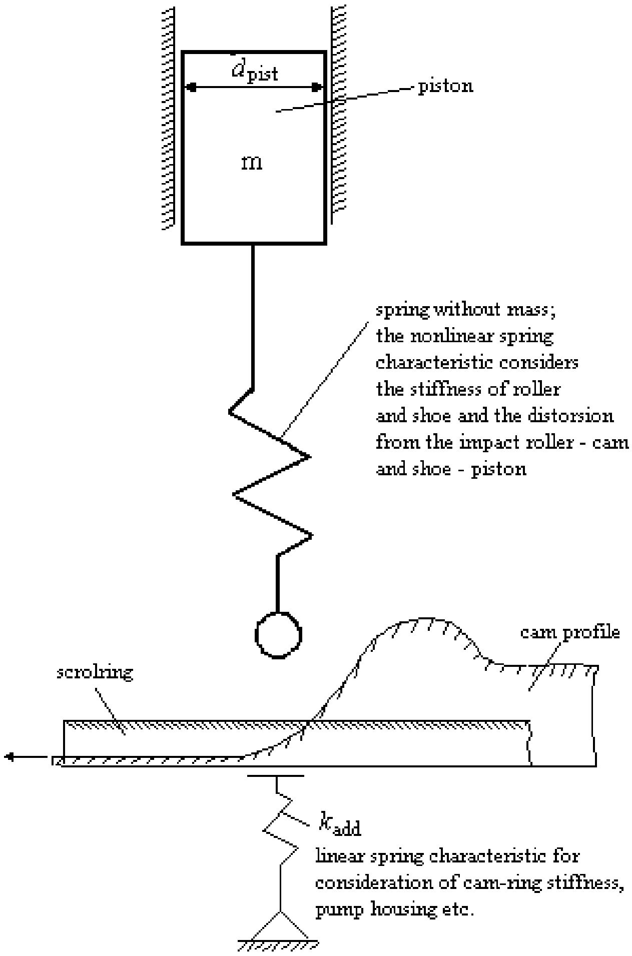


Figure 98: Scrolring-Roller and Shoe-Piston Connection

12.1.2. Input Parameters

Description of input data for the **Radial Piston Distributor Manual**:

Element name	The name of the element is specified as default.
Pressure in cam chamber	Specify pressure in Figure 99).
Radius of cam base circle	Specify the radius of cam base circle R_{base}. (refer to Figure 99 and Figure 100)
Radius of cam scrolring	Specify radius of the scrolring R_{scrol} (refer to Figure 99)
ROLLER DATA	
Radius	Specify radius of roller r_{roll}. (refer to Figure 100)
Effective width	Specify effective width of roller b_{roll}. (refer to Figure 99)
PISTON DATA	
Mass	Specify mass *m* of one piston.
Diameter	Specify piston diameter d_{pist}. (refer to Figure 98 and Figure 99)
Friction force	Specify constant Coulomb friction force (if any) at piston motion.
Number of pistons	Specify the number of pistons.
DISTANCE FROM PISTON CENTER OF GRAVITY	
to roller end	Specify distance between roller end and piston's center of gravity l_{cogr}. (refer to Figure 99)
to base circle	Specify start position of piston X_{start} , which is the distance between cam base circle and piston's center of gravity at calculation start (refer to Figure 99)
SCROLRING-ROLLER & SHOE-PISTON IMPACT	
According to the data below, BOOST Hydsim will simulate the spring stiffness between piston and cam profile. Elastic contact between piston and shoe and between roller and cam profile will be replaced with this spring. BOOST Hydsim will calculate characteristic of the spring, which represents a series of connections (piston-shoe-roller-cam profile).	
Impact factor	Specify impact factor ε which considers the impact forces between roller shoe and piston as well as roller and cam profile: $$F_{impact} = k_{FC} \Delta x \cdot \varepsilon^2 \qquad (207)$$ where: F_{impact} impact force k_{FC} spring stiffness Δx spring compression

Shoe radius	Specify radius of spherical surface of roller shoe R_{shoe} at its arc of contact with piston. (refer to Figure 99)
Stiffness of serial spring to piston	Specify stiffness of serial spring k_{add} along piston axis (for consideration of cam-ring stiffness, pump housing, etc.). If the stiffness is not specified or set to 0, no serial spring is considered (infinite stiffness is assumed). Refer to Figure 98 for more information. If stiffness of serial spring is defined ($k_{add} > 0$), then equivalent stiffness is given by: $$k = \frac{k_{FC} k_{add}}{(k_{FC} + k_{add})} \quad (208)$$ else $k = k_{FC}$
Young's modulus of pump material	Specify modulus of elasticity of pump material.
Poisson's ratio of pump material	Enter Poisson's ratio of pump material.
Transmission ratio vs. Reference speed	Specify transmission ratio between **RPD pump** rotation speed and Reference speed in **Calculation Control** dialog. Pump speed is calculated as Reference speed multiplied with transmission ratio. **Note:** If an input connection in *w*-direction exists (from e.g. **Shaft**), then Transmission ratio input is disabled. In this case connected element (e.g. **Shaft**) speed is used as **RPD pump** speed.
CAM PROFILE	
Shift angle relative to calculation start [deg]	Not active with this element.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Cam profile lift in radial direction (x)	Click to activate lift of reciprocating cam follower in 2nd column.
Cam radial acceleration at 1000 rpm	Click to activate follower acceleration at 1000 rpm in 2nd column
Cam profile table	Cam profile has to be specified for the entire calculation interval (End Reference angle) given in the **Calculation Control** dialog. However, if more than one revolution of the cam is considered in the calculation, it is sufficient to define the cam profile for one revolution of the camshaft (360 degrees). Intermediate points of the cam profile are calculated by linear interpolation.

☞ **Note:** Cam lift as input data must be used with extreme caution. If the lift data are not precise enough, the numerical derivatives (velocity and acceleration) may contain coarse errors, and calculation results will be incorrect. The cam acceleration curve needs to be checked carefully within each calculation. Intermediate points of the cam profile are calculated by linear interpolation if necessary.
Cam profiles have to be specified for the same interval of Reference angles as given in the **Calculation Control** dialog box. However, if more than one revolution of the cam is considered in the calculation, it is sufficient to define the cam profile for one revolution of the pump rotor. (360 degrees).Cam profile can be viewed in Cartesian coordinates by pressing the **View** bar. If pressed, an IMPRESS Chart window is opened where the cam profile lift/acceleration in radial direction is displayed.

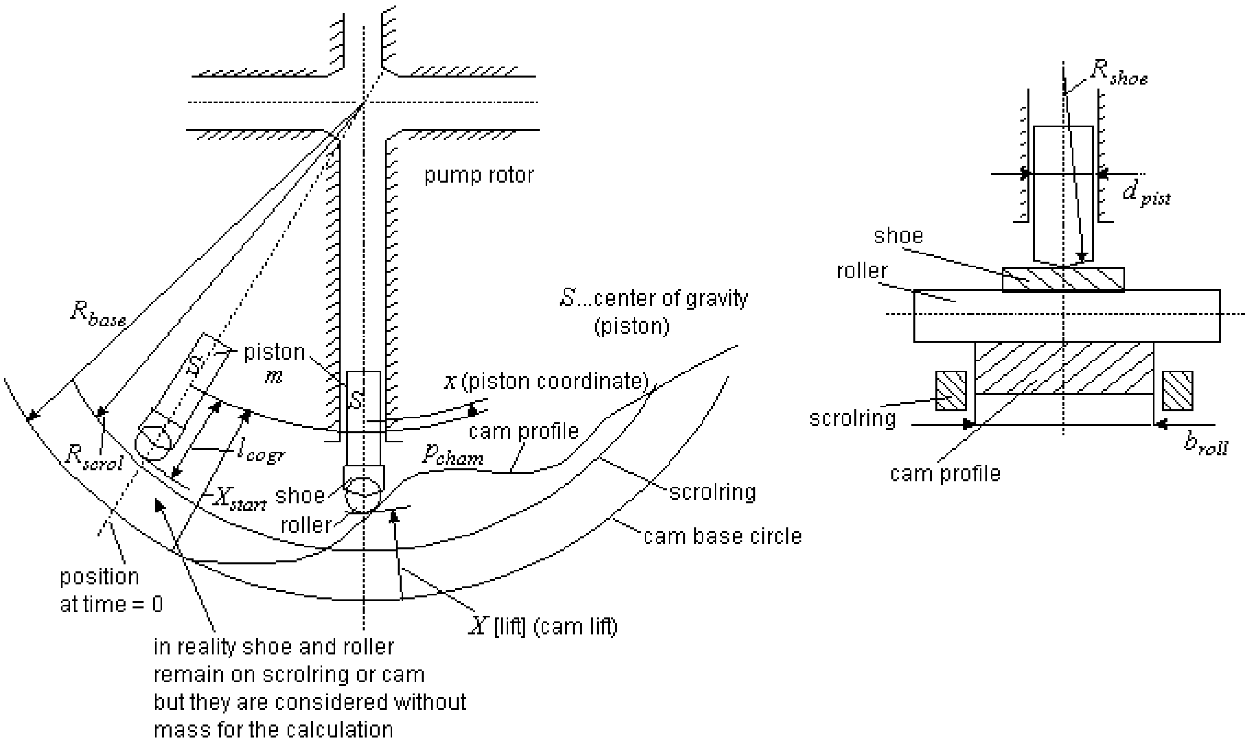


Figure 99: Pump Geometry

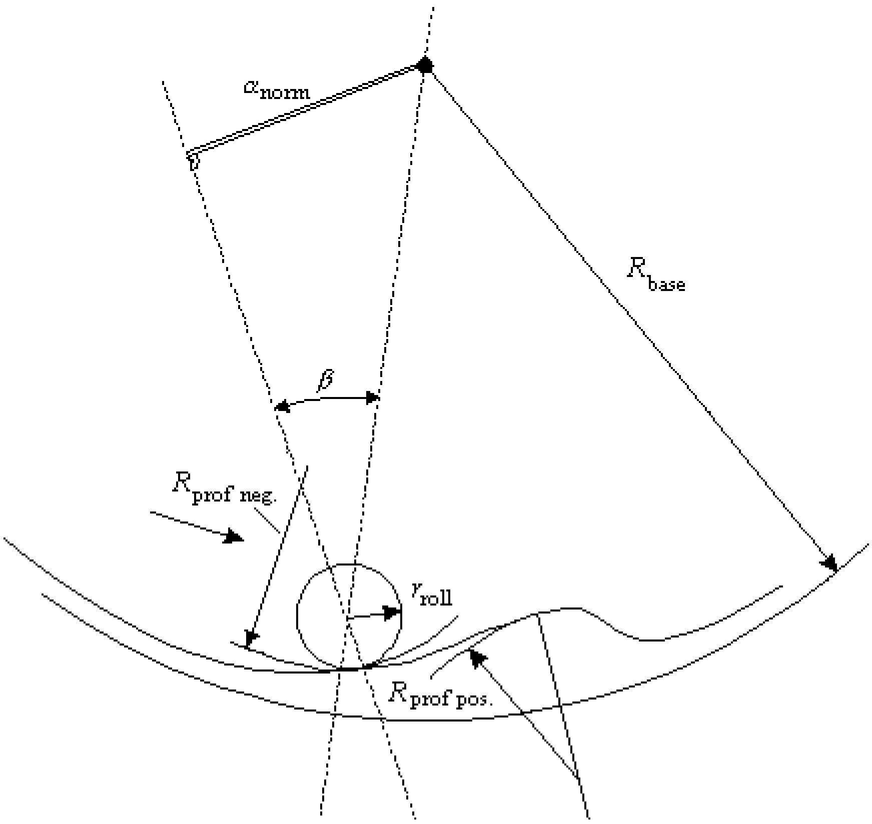


Figure 100: Cam Profile Geometry

12.1.3. Initial Conditions

Description of initial conditions:

INITIAL PISTON TRANSLATION	
x-coordinate	Specify initial *x*-position of piston.
velocity in x-direction	Specify initial velocity of piston in *x*-direction.
INITIAL PUMP ROTATION	
w-angle	Specify initial position of pump rotor in *w*-direction.
angular velocity in w-direction	Specify initial angular velocity of pump rotor in *w*-direction.

All initial values that are not specified under initials are set to 0.

12.1.4. Modify Parameter

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

piston mass:	SI Unit:	kg
piston diameter:	SI Unit:	m
Coulomb friction force at piston:	SI Unit:	N
pressure in cam chamber:	SI Unit:	Pa
impact factor (velocity ratio):	SI Unit:	---
effective width of roller:	SI Unit:	m
contact surface radius of roller shoe:	SI Unit:	m
stiffness of serial spring to piston	SI Unit:	N/m

12.1.5. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

rotation angle of pump rotor (input end)	*w*-coordinate (rotation angle) of RPD pump rotor. If **RPD Pump** is a boundary element (i.e. an input mechanical connection in *w*-direction does not exist), rotation angle of pump rotor is the product of Reference speed $\dot{\varphi}_{ref}$, transmission ratio i_{tr} and time interval from the beginning of calculation t: $$w = \dot{\varphi}_{pump}\, t = i_{tr}\dot{\varphi}_{ref}\, t\,. \quad (209)$$ If an input mechanical connection in *w*-direction exists, rotation angle of pump rotor is calculated from the motion of the **Shaft** element connected to **RPD Pump**.
angular velocity of pump rotor (input end)	Angular velocity in *w*-direction of pump rotor. If **RPD Pump** is a boundary element, its angular velocity is equal to Reference speed multiplied by transmission ratio. Otherwise (input mechanical connection in *w*-direction exists) angular velocity is calculated from the velocity of the connected **Shaft** element.
lift of cam profile in radial direction	Lift of cam profile is taken from lift table of the profile in **Input Data** dialog box or calculated by integration in time domain from cam acceleration table.
profile velocity in radial direction at 1000 rpm	Profile velocity in radial direction at 1000 rpm is derived in time domain from cam lift or integrated from acceleration defined in **Input Data** dialog box.
profile acceleration in radial direction at 1000 rpm	Profile acceleration in radial-direction at 1000 rpm is taken from cam acceleration table in **Input Data** dialog box or derived in time domain from cam lift table.
pressure angle of cam profile	Pressure angle of cam profile corresponds to β in Figure 100.
arm of the force normal to cam profile	Arm of the force normal to cam profile corresponds to a_{norm} in Figure 100.

torsional distortion angle of pump rotor	Torsional distortion angle of rotor is the difference between actual rotation angle and Reference angle which is calculated from the Reference speed. It is nonzero only if an input mechanical connection in w-direction exists.
force normal to cam profile	Resistance force normal to cam profile F_{norm} .It is calculated from piston reaction force in radial direction F_x by the formula: $$F_{norm} = \frac{F_x}{cos\,\beta} \quad (210)$$ where β is pressure angle.
torque of reaction force about axis w (drive torque)	Cam body torque caused by force F_{norm} which acts at an angle β to *x*-direction: $$T_{cp} = F_{norm} sin\beta \cdot (R_{base} - X_{lift} - r_{roll}) \quad (211)$$ where X_{lift} is cam profile lift in *x*-direction. Refer to Figure 3 for more information.
resultant force acting on piston	Resultant force of piston element is sum of all connections and external forces acting on it. They are: damping and stiffness forces, Coulomb friction forces, centrifugal forces and hydraulic connection forces. $$F_R = -F_d - F_k - F_{fric} + F_\omega + F_p$$ (refer to Section 12.1.6, Equation 212)
hydraulic force acting on piston	Hydraulic force F_p is sum of pressure forces acting on input and output piston areas (refer to Section 12.1.6, Equation 212)
centrifugal force acting on piston	Piston centrifugal inertia force F_ω (refer to Section 12.1.6, Equation 212)
impact force at scrolring-roller and shoe-piston contact	Impact stiffness force F_k (refer to Section 12.1.6, Equation 212)
viscous shear force from leakage through piston	Shear force F_{shear} is calculated in **Leakage | Annular Gap** element which is connected to piston element via special connection (refer to Section 12.1.6, Equation 212)

12.1.6. Mechanical Model of RPD Pump

Differential Equation of Single Mass Oscillator

The system in Figure 101 (piston, shoe, and roller model) can be represented by a single-mass oscillator. Differences in simulation results between this system and two mass - oscillator with constant pressure and same initial conditions are negligible, and the impact phenomena inside this system can be modeled with a single-mass system.

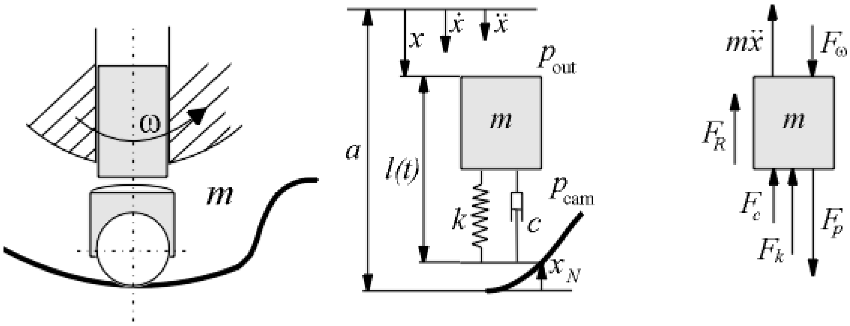


Figure 101: Equivalent Dynamic Model of Piston, Roller and Shoe

In single mass system, piston, shoe, and roller together form a single rigid body. Differential equation of motion for this body is as follows:

$$m\ddot{x} = -F_d - F_{shear} - F_k - F_R + F_\omega + F_p, \tag{212}$$

where:

F_d	....	$d(\dot{x}_N - \dot{x})$ viscous damping force
F_{shear}	....	$\frac{\pi}{4} d_p (p_{in} - p_{out})(d_b - d_p) - 2\pi\mu \frac{d_p}{d_b - d_p} L_{gap}\dot{x}$ (refer to Chapter 11)
F_k	....	$k(l_0 - (\ell_{cogr} - x_N - x))$ stiffness force (for k refer to Equation 208)
F_R	....	Coulomb friction force
F_ω	....	$mr\omega^2$ with $r(x) = r_0 + x = R_{base} - X_{start} + x$ (centrifugal force)
F_p	....	$A(p_{out} - p_{cam})$ resultant hydraulic force

For the calculation of contact forces, the Hertz contact theory is applied. It is based on the assumption that body is isotropic and linear elastic, and that the dimension of the contact area is very small in comparison with dimensions of contact bodies. In **RPD Pump** model, there are two contact cases:

1. **Contact between piston and roller shoe**. This can be modeled with a sphere in contact with a plate. Stiffness between these two bodies in contact can be defined as a function of Δx_{s_p}, where Δx_{s_p} represents deformation of bodies after their first contact:

$$k_{s\_p} = \sqrt{\Delta x_{s\_p}} \frac{E\sqrt{R_{shoe}}}{(1-\nu^2)}, \tag{213}$$

where E is Young's modulus of pump material (assumed the same for piston, roller, shoe and scrolring) and ν is the Poisson's ratio ($\nu = 0.3$ here).

2. **Contact between roller and cam profile**. It can be modelled with cylinder in contact with plate. Stiffness between these two bodies in contact can be defined as function of Δx_{r_c}, where Δx_{r_c} represents further closing of bodies after first contact.

$$k_{r\_c} = \sqrt[3]{\Delta x_{r\_c}}\,\frac{E\sqrt[3]{b_{roll} r_{roll}}}{\left(1-\nu^2\right)\pi}\,\frac{32}{9\sqrt[3]{3}}. \tag{214}$$

For a single-mass oscillator, two springs are connected in series with an assumption that both deformations occur at the same time. Equivalent spring stiffness k is given by:

$$\frac{1}{k} = \frac{1}{k_{s\_p}} + \frac{1}{k_{r\_c}}. \tag{215}$$

Substitution of Equations 213 and 214 into 215 yields:

$$k_{FC} = \frac{E}{\left(1-\nu^2\right)}\,\frac{\sqrt{\Delta x \cdot R_{shoe}}}{1+\dfrac{9\pi\sqrt[3]{3}}{32}\sqrt[6]{\dfrac{\left(\Delta x \cdot R_{shoe}^3\right)}{b_{roll}^2 r_{roll}^2}}}. \tag{216}$$

Nonlinear inhomogeneous differential equation of piston motion (Equation 212) can be written in the form:

$$m\ddot{x} + \varepsilon^2 k_{FC}\Delta x - mx\omega^2 = -F_R + mr_0\omega^2 + A\left(p_{out} - p_{cam}\right) \tag{217}$$

where:

$$\Delta x = \ell_o - \ell_{cogr} + x_N + x$$

$$r_o = R_{base} - X_{start}$$

Impact factor:

$\varepsilon^2 = 1$ .... during the compression period

$0 < \varepsilon^2 < 1$ ...... during the restitution period

Note that in Equation 217, Δx can only be a positive value that implies $(\ell_{cogr} - x_N) < (\ell_0 + x)$. If Δx is negative, there is no impact and the term $\varepsilon^2 k\Delta x$ in Equation 217 vanishes.

12.1.6.1. Definition of Impact Factor

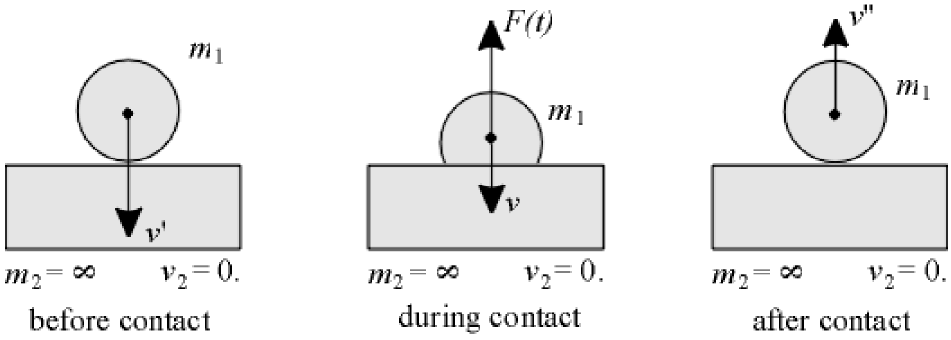


Figure 102: Contact Deformation

Motion of mass m_1 during contact is governed by the following equation:

$$m_1 \dot{v} = -F(t) \tag{218}$$

Contact force $F(t)$ between finite mass m_1 and infinite mass m_2 acts in small time interval $\Delta t = t'' - t'$, where t' is the start of contact and t'' is the end of contact..

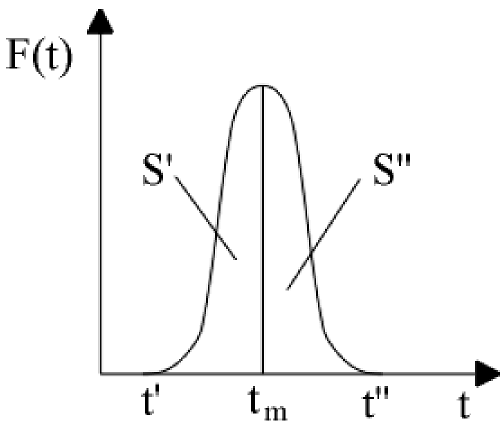


Figure 103: Contact force

Equation 218 can be rearranged as follows:

Compression phase $$m_1 v_m - m_1 v' = -\int_{t'}^{t_m} F dt = -S' , \tag{219}$$

Restitution phase $$m_1 v'' - m_1 v_m = -\int_{t_m}^{t''} F dt = -S'' \tag{220}$$

Where t_m is the time instant at which velocity v_m is 0. Rearranging terms, we obtain:

$$v' = \frac{S'}{m_1} \tag{221}$$

In Equations 219 and 220, areas S' and S'' are the contact force impulses in compression and restitution phases, respectively.

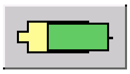
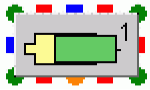
Ratio $\frac{S''}{S'}$ depends only on material characteristics of the contact elements.

Impact factor is defined as velocity ratio before and after impact:

$$\varepsilon = \frac{v''}{v'} = \frac{S''}{S'}$$

$\varepsilon = 1$ for an ideal elastic contact,
$\varepsilon = 0$ for an ideal plastic contact.

12.2. Plunger

Element Name:	**Plunger**	
Element Icon:		
Definition:	This element serves to define the Plunger element.	
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 4 wire pins: 1	

Note: Rigid **Plunger** element has only one degree-of-freedom (translation in x direction. Elastic **Plunger** has two DOFs: translation along x axis for input and output end, separately.

Note: Hydraulic connections can be defined only in local x-direction. **Plunger** may have only one hydraulic connection. On output end (pump chamber) one and only one hydraulic connection must be specified.

Note: Mechanical connections can be defined only in local x-direction of connection. On input end, a mechanical connection to e.g. **Cam Profile** element has to be defined (refer to Figure 104).

Note: For **Plunger** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: Local x axis of **Plunger** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter: Direction of Element motion.

Note: Standard connections can be defined in x-direction only. On input end, a mechanical connection to e.g. **Cam Profile** element has to be defined (refer to Figure 104). On output end (pump chamber) one and only one hydraulic connection must be specified. Multiple mechanical connections to the same element are not allowed (all connections except the first will be ignored).
If the **Plunger** has a metering ramp, it has to be connected (via special connection) either to a **In-line Spill Port** or a **Distributor Spill Port** element. In this case, the geometry of the plunger has to be specified in the respective input dialogs of the connected elements of the **Port** group (refer to Chapter 12 for more information). Rigid **Plunger** element has only one degree-of-freedom (translation in x direction). Elastic **Plunger** has two DOFs: translation along x axis for input and output end, separately.

The mechanical model of the **Plunger** is shown in Figure 104.

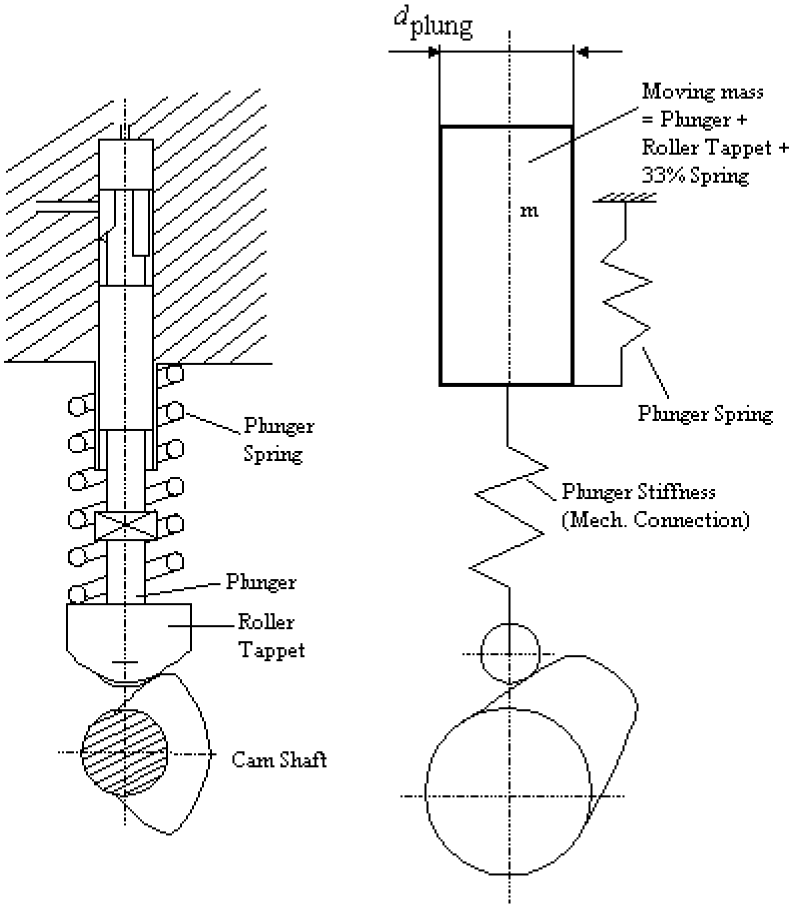


Figure 104: Mechanical Model of Plunger

12.2.1. Input Parameters

Description of input data for the **Plunger**:

Element name	The name of the element is specified as default.
Rigid Body	**Rigid Plunger** is a rigid body with 1 DOF, i.e. without elasticity between the input and output end. Coordinates and velocities on input and output end of **Plunger** element are same.
Moving mass	Specify moving mass *m*. It is equal to the plunger mass plus the roller and tappet mass plus 33% of the total mass of the attached springs (connections).
Coulomb friction force	Specify the Coulomb friction force in local x-direction.
Elastic Body	**Elastic Plunger** implies an elastic body with the axial elasticity between the input and output end. Stiffness and damping characteristics of **Elastic Plunger** are calculated according to the geometric and material data.
Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the **Elastic Plunger** (refer to **6.2.2**).

Plunger diameter	Specify diameter of plunger d_{plung} (refer to Figure 104). Plunger input/output area (calculated from diameter) will be subjected to hydraulic pressure. This will produce hydraulic connection forces which will enter dynamical equation of motion of **Plunger** as additional forces (Equation of Motion). Furthermore, plunger velocity will produce flow rate which will enter continuity equation of attached **Volume** element (refer to Chapter 8 for more information).
Pressure in cam chamber	Specify pressure in cam chamber. Pressure will act on input side of plunger. It may not be specified if input hydraulic connection exists.
PLUNGER SPRING	
Preload	Specify preload force of **Plunger** spring (at cam base circle).
Stiffness	Specify stiffness of **Plunger** spring.
Damping	Specify damping coefficient of **Plunger** spring.
DIRECTION OF ELEMENT MOTION	
x(e1) **y(e2)** **z(e3)**	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated.	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is result of plunger spring deformation ($c_0\dot{x} + k_0 x + F_0$).	

12.2.2. Initial Conditions

Initial Conditions of Plunger are identical to the **Initial Conditions of Standard Piston** (refer to Section **7.1.3**).

12.2.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar. Modifiable parameters:

moving mass	SI Unit:	kg
plunger diameter	SI Unit:	m
Coulomb friction force	SI Unit:	N
pressure in cam chamber	SI Unit:	Pa (N/m^2)
preload of plunger spring	SI Unit:	N
stiffness of plunger spring	SI Unit:	N/m
damping of plunger spring	SI Unit:	Ns/m

12.2.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the check box in front of output parameter must be pressed. Description of output parameters:

x-coordinate of plunger (input side)	Local x-coordinate of input side of **Plunger.**
plunger velocity in x-direction (input side)	Velocity in local x-direction of input side of **Plunger.**
plunger acceleration in x-direction (input side)	Acceleration in local x-direction of input side of **Plunger**.
x-coordinate of plunger (output side)	Local x-coordinate of output side of **Plunger**.
plunger velocity in x-direction (output side)	Velocity in local x-direction of output side of **Plunger**.
plunger acceleration in x-direction (output side)	Acceleration in local x-direction of output side of **Plunger.**
Forces are calculated in local x-direction:	
driving plunger force (mechanical)	Sum of all mechanical input forces and friction force.
work of driving force	Product of driving force and **Plunger** displacement.

12.2.5. Equation of Motion of Rigid Plunger

Motion of **Rigid Plunger** in local coordinate system is governed by the following equation:

$$
\begin{aligned}
m\ddot{x} + c_0\dot{x} + k_0x + \sum_{i=1}^{n} c_i(\dot{x}-\dot{x}_i) - \sum_{j=1}^{l} c_j(\dot{x}-\dot{x}_j) + \sum_{i=1}^{n} k_i(x-x_i) - \sum_{j=1}^{l} k_j(x-x_j) = \\
-F_0 + \sum_{i=1}^{n} F_{0i} - \sum_{j=1}^{l} F_{0j} - F_{hyd} - F_{frict} - F_{shear}
\end{aligned}
\tag{222}
$$

where:

m	Plunger mass
x	local x-coordinate of plunger
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends
Forces on right side of equation are defined in local x-direction:	
F_{hyd}	Hydraulic resistance force
F_{frict}	Coulomb friction force
F_{shear}	Viscous friction force due to leakage (if any)

Hydraulic force F_{hyd} is calculated from the equation:

$$F_{hyd} = \left(p_{out} - p_{in} - p_{cam}\right)\frac{\pi}{4} d_{plung}^2 , \tag{223}$$

where :

p_{in} and p_{out}	Input and output pressures
p_{cam}	Pressure in cam chamber (on input end)
d_{plung}	Plunger diameter

Damping and stiffness constants, preload of plunger spring and pressure in cam chamber have to be specified directly in the input dialog of the **Plunger**.

Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or Hydraulic Boundaries). Note that if the pressure in cam chamber is constant (usual case), it is enough to define p_{cam} in the input dialog box and no input hydraulic connection is necessary (p_{in}=0).

Viscous friction (shear) force F_{shear} is included only if a **Leakage** element is attached (by special connection) to **Plunger**. Force F_{shear} is calculated from the Equation 236 (refer to Section 13.1.5.1).

Equation of Motion of Plunger (Equation 222) is very much similar to the Equation of Motion of Standard Piston (Equation **62**). In essence, the dynamic models of **Plunger** and **Piston** elements are the same. The only difference is that **Plunger** contains an additional spring and does not have an input and output stop.

12.2.6. Equation of Motion of Elastic Plunger

Motion of **Plunger with Elastic Body** is governed by the following equations:

$$\begin{aligned}
&(m_{inp} + m_{ad_inp})\ddot{x}_{inp} + c_0\dot{x}_{inp} + k_0 x_{inp} + \sum_{i=1}^{n} c_i(\dot{x}_{inp} - \dot{x}_i) + \sum_{i=1}^{n} k_i(x_{inp} - x_i) = \\
&- F_0 + \sum_{i=1}^{n} F_{0i} + F_{hyd_inp} - F_{C_inp} - 0.5F_{shear} + F_{stiff} + F_{damp}, \\
&(m_{out} + m_{ad_out})\ddot{x}_{out} - \sum_{j=1}^{l} c_j(\dot{x}_{out} - \dot{x}_j) - \sum_{j=1}^{l} k_j(x_{out} - x_j) = \\
&- \sum_{j=1}^{l} F_{0j} + F_{hyd_out} - F_{C_out} - 0.5F_{shear} - F_{stiff} - F_{damp}, \\
&F_{stiff} = k_{ps}(x_{inp} - x_{out}), \\
&F_{damp} = c_{ps}(\dot{x}_{inp} - \dot{x}_{out})
\end{aligned} \tag{224}$$

Masses m_{inp} and m_{out} are calculated according to the position of plunger gravity center ($m_{inp}+m_{out}=m$).

Hydraulic forces F_{hyd_inp} and F_{hyd_out} are calculated by:

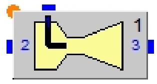
$$F_{hyd\_inp} = -(p_{in} + p_{cam})\frac{\pi}{4}d_{plung}^2, \tag{225}$$

$$F_{hyd\_out} = p_{out}\frac{\pi}{4}d_{plung}^2, \tag{226}$$

where p_{inp} is the input pressure, p_{cam} is the pressure in cam chamber, p_{out} is the output pressure and d_{plung} is the plunger diameter.

☞ **Note:** Viscous friction (shear) force Fshear is split into two same parts on input and output end of piston.

12.3. Hydraulic Ejector

Element Name:	**Ejector (jet pump)**
Element Icon:	
Definition:	This element serves to model an ejector device in which a high-pressure liquid jet is used to drive low-pressure fluid.
Connecting pins:	standard pins: 3 (hydraulic) special pins: 0 wire pins: 1 All three hydraulic connections must be specified. 

Note: Flow rates through connections may be only positive.

Note: Ejector is designed to operate at a single optimum point. Deviation from this optimum results in a dramatic deterioration of the ejector performance.

12.3.1. Input Parameters

Description of input data for the **Ejector**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Type: Program-calculated	1-D fluid flow in the ejector is governed by conservation equations of continuity, momentum, and energy.
Ejector efficiency	η_{ef} ejector efficiency is the ratio of the suction mass flow rate divided by the driving mass flow.
Cross-sectional areas	
at Connections	
Driving inlet (1)	Specify cross-sectional area at driving flow inlet A_i.
Suction inlet (2)	Specify cross-sectional area at suction flow inlet A_s.
Diffuser exit (3)	Specify cross-sectional area at diffuser flow outlet A_d.
at Mixing section	
Driving flow	Specify cross-sectional area at driving flow inlet into mixing section A_1.
Suction flow	Specify cross-sectional area at suction flow inlet into mixing section A_2.
Diffuser flow	Specify cross-sectional area at diffuser flow outlet from mixing section A_3.
Flow	

Flow Resistances	
Driving flow	Specify flow resistance of driving flow pipe ζ_1.
Suction flow	Specify flow resistance of suction flow pipe ζ_2.
Diffuser flow	Specify flow resistance of diffuser flow pipe ζ_d.
Diffuser	
Recovery efficiency	Specify η_d pressure recovery efficiency of diffuser pipe

12.3.2. Initial Conditions

Initial conditions cannot be specified for a **Ejector** element.

12.3.3. Modify Parameter

Ejector element has no modifiable parameters.

12.3.4. Output Parameters

Path: **Element** | **Store Results**

The **Output Parameters Dialog Box** of **Ejector** is shown in Figure 105.

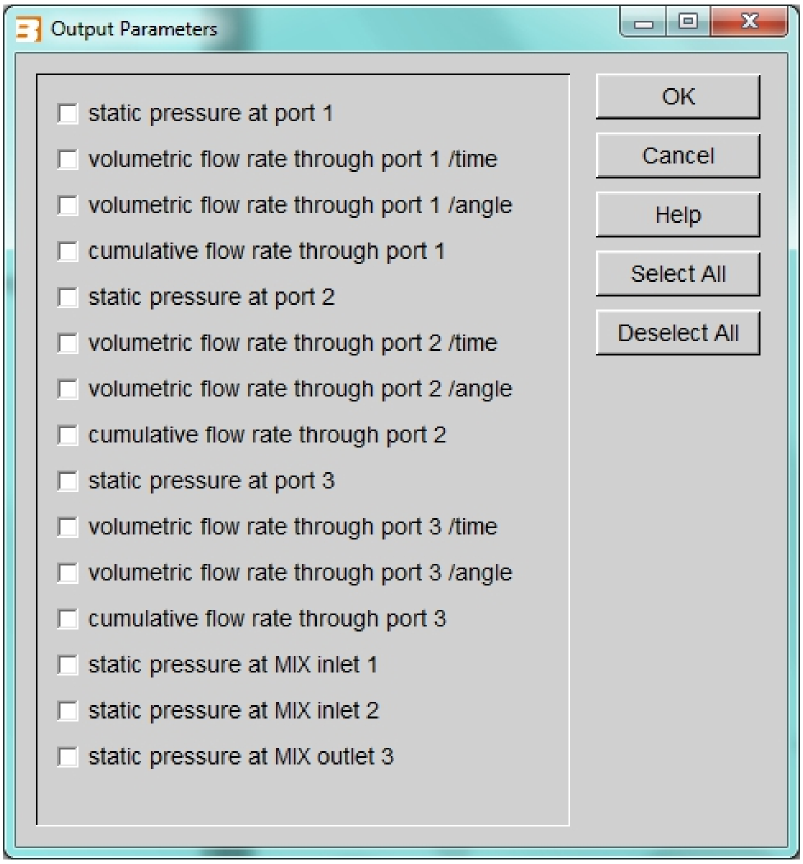


Figure 105: Output Parameters of Ejector Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

12.3.5. Ejector Model

The operation principle of **Hydraulic Ejector** (also called **jet pump**) is based on the conversion of the pressure energy of the driving (primary) fluid into the flow energy (velocity) through driving nozzle. The resultant jet of high velocity creates a low pressure area in the suction chamber causing the suction of secondary fluid into this chamber. Consequently, there is an exchange of momentum between the two streams in the mixing chamber resulting in a uniformly mixed stream traveling at an intermediate velocity between the driving and suction velocities of the fluid. The diffuser is shaped to convert the kinetic energy of the mixture to pressure rise at the discharge flange with a minimum energy loss. The absence of moving mechanical parts eliminates the potential problems associated with bearings, seals and lubrication. Therefore such pumps are widely used because of their simplicity and high reliability [25].

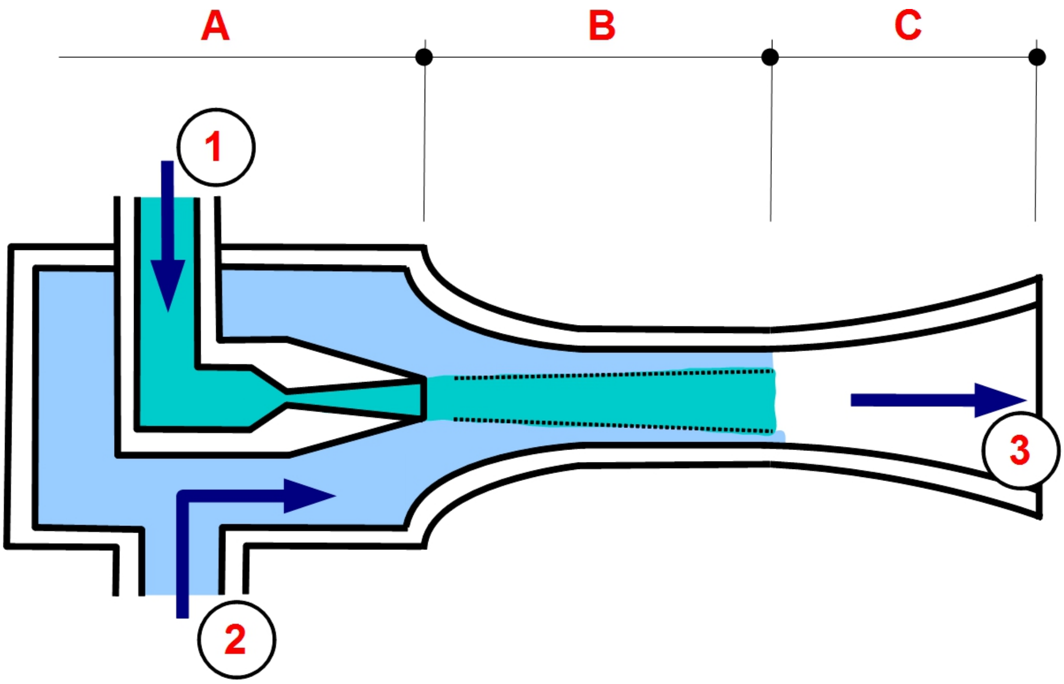


Figure 106: Schematic of Hydraulic Ejector (jet pump)

A schematic diagram of typical **Hydraulic Ejector** is shown in Figure 106. The ejector may be divided in three pieces:

A. **Injector** which is composed of a converging nozzle, and an entry for the injected fluid,
B. **Mixture zone** which is usually a wider converging nozzle (to accommodate the two fluids), or/and,
C. **Diffuser** which is a diverging nozzle used to be called the delivery nozzle.

Figure 107 illustrates the typical pressure variation along the ejector.

[25] Hammoud, A.H. Effect of design and operational parameters on jet pump performance *Proceedings of the 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece, August 21-23, 2006 (pp245-252)*

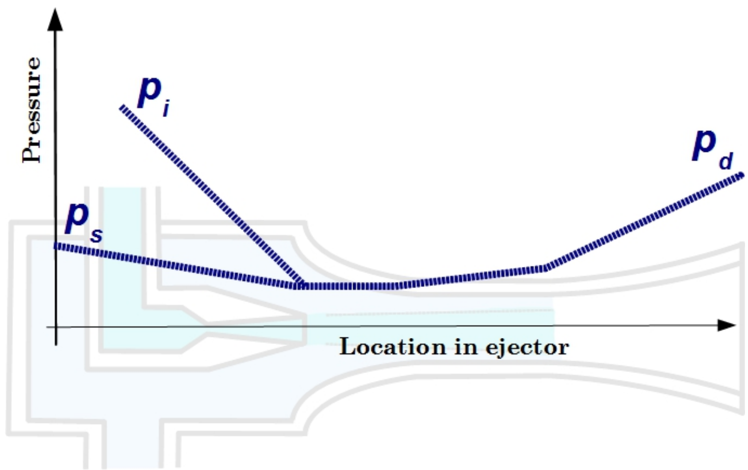


Figure 107: Pressure variation along the ejector

The final pressure at the outlet of the ejector will depend principally upon the following circumstances: pressure and mass flow rate of the driving flow, pressure and mass flow rate of the suction flow and pressure losses due to friction that the two fluids will experience inside it.
For simplicity the one-dimensional theory is applied here for the ejector modeling.

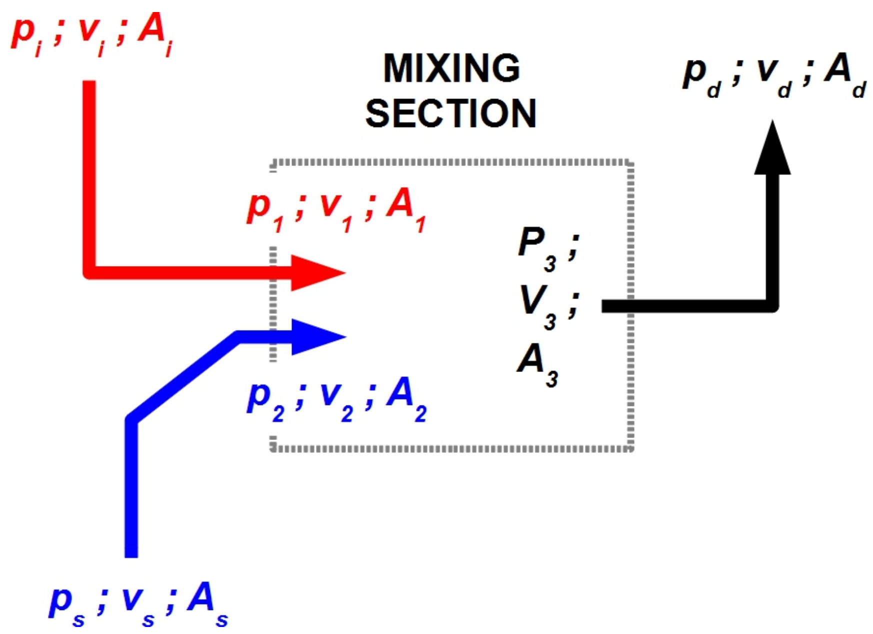


Figure 108: Ejector mixing section

The following notations are used in Figure 108:

Port 1

p_i	-	pressure at driving flow inlet
v_i	-	flow velocity at driving flow inlet
A_i	-	cross-sectional area at driving flow inlet

Port 2

p_s	-	pressure at suction flow inlet
v_s	-	flow velocity at suction flow inlet
A_s	-	cross-sectional area at suction flow inlet

Port 3

p_d - pressure at diffuser flow outlet

v_d - flow velocity at diffuser flow outlet

A_d - cross-sectional area at diffuser flow outlet

Figure 108 shows the operational principle of the ejector. Using notation in the figure and letting $\zeta_1, \zeta_2, \zeta_d$ be head loss of each duct and η_d pressure recovery efficiency of the diffuser pipe following equations are satisfied:

$$\frac{p_i}{\gamma_i} + \frac{v_i^2}{2g} = \frac{p_1}{\gamma_1} + (1+\zeta_1)\frac{v_1^2}{2g}$$

$$\frac{p_s}{\gamma_s} + \frac{v_s^2}{2g} = \frac{p_2}{\gamma_2} + (1+\zeta_2)\frac{v_2^2}{2g}$$

$$\frac{p_3}{\gamma_3} + \frac{v_3^2}{2g} = \frac{p_d}{\gamma_d} + (1+\zeta_d)\frac{v_d^2}{2g} + (1-\eta_d)\left(\frac{v_3^2}{2g} - \frac{v_d^2}{2g}\right)$$

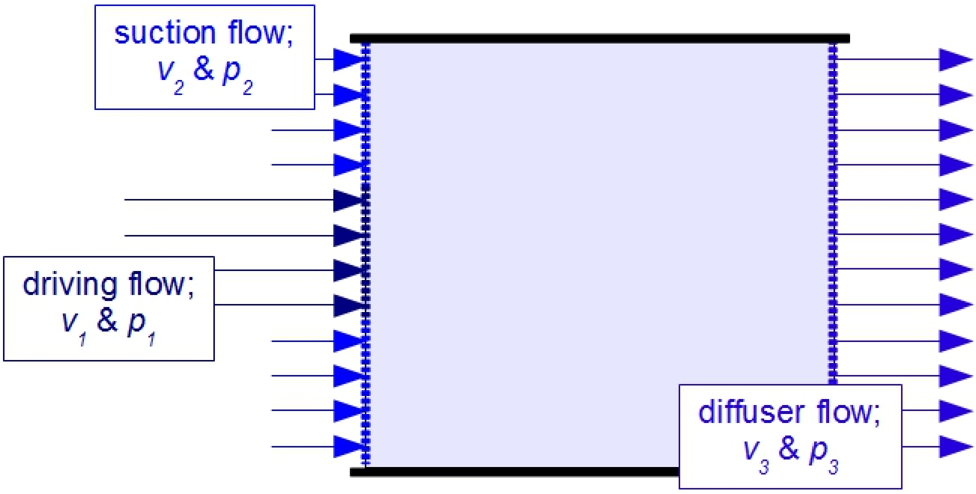


Figure 109: Mixture zone with driving, suction and diffuser flows

As illustrated in Figure 109, the driving flow with high velocity draws the suction flow into the mixing section and diffuser. The drawing process is a result of friction and shear stresses at the interface between the two streams. The turbulent shear stresses created by the driving and suction flows of different velocities initiate the mixing process and exchange of momentum. The following conservation laws of mass and momentum apply:

$$p_3 A_3 - p_2 A_2 - p_1 A_1 = A_1 \rho_1 v_1^2 + A_2 \rho_2 v_2^2 - A_3 \rho_3 v_3^2$$

$$A_1 \rho_1 v_1 + A_2 \rho_2 v_2 = A_3 \rho_3 v_3$$

$$A_i \rho_i v_i = A_1 \rho_1 v_1$$

$$A_s \rho_s v_s = A_2 \rho_2 v_2$$

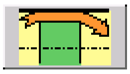
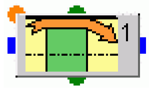
$$A_d \rho_d v_d = A_3 \rho_3 v_3$$

The ejector efficiency can be derived from the fluid dynamics equations of compressible flow and is a product of the frictional losses in the nozzle, mixing section and diffuser area. In the conventional ejector analysis it is common to define ejector efficiency as the ratio of the mass rate of the suction flow divided by the mas rate of the driving flow:

$$\eta_{ef} = \frac{\dot{m}_s}{\dot{m}_i} = \frac{A_s \rho_s v_s}{A_i \rho_i v_i}$$

13. LEAKAGE

13.1. Annular Gap

Element Name:	**Annular Gap**	
Element Icon:		
Definition:	This element serves to model the fluid leakage through the annular gap between piston-type element (**Piston**, **Plunger, Needle** guide, etc.) and barrel (cylinder wall).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection can be specified.	

Note: **Annular Gap** element takes information from piston-type elements about positions and velocities via special connections.
One **Annular Gap** element can account for the leakage for several piston-type elements of the same dimension and equal velocities.

Note: **Annular Gap** element calculates not only the flow through piston cylinder gap but also the viscous shear force exerted by this flow on piston.

Note: If **Annular Gap** element is connected via two special connections to the two piston-type elements, then leakage is calculated in the gap between these two pistons, i.e. the leakage gap length is equal to the lift difference between the pistons.

13.1.1. Input Parameters

Description of input data for the **Annular Gap**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Piston diameter	Specify diameter of piston-type element. **Note:** If check box is off, **Annular Gap** element takes the piston diameter from one of the piston-type elements connected by special connection which has smaller diameter.
Number of pistons	Specify number of piston-type elements. One **Leakage** element can account for the leakage of several pistons (plungers) of the same dimension.

Initial gap length	Specify initial length of annular gap.
GAP LENGTH	Switches between **Constant** gap length (initial value) and **Variable** gap length (initial gap + plunger lift). If plunger contains a leakage groove and leakage path always stays open (as shown in Figure 110), gap length is constant. If leakage path depends on plunger position, variable gap length has to be used. In this case initial gap L_{init} must be specified. The actual gap L_{gap} length is calculated from the formula: $L_{gap} = L_{init} + (x_p - x_b)$ where x_p is the piston lift and x_b is barrel lift. Variable gap length might be required for mechanical injection pumps. Note that initial gap length L_{init} has to be negative for the case if the positive plunger motion reduces the gap length.

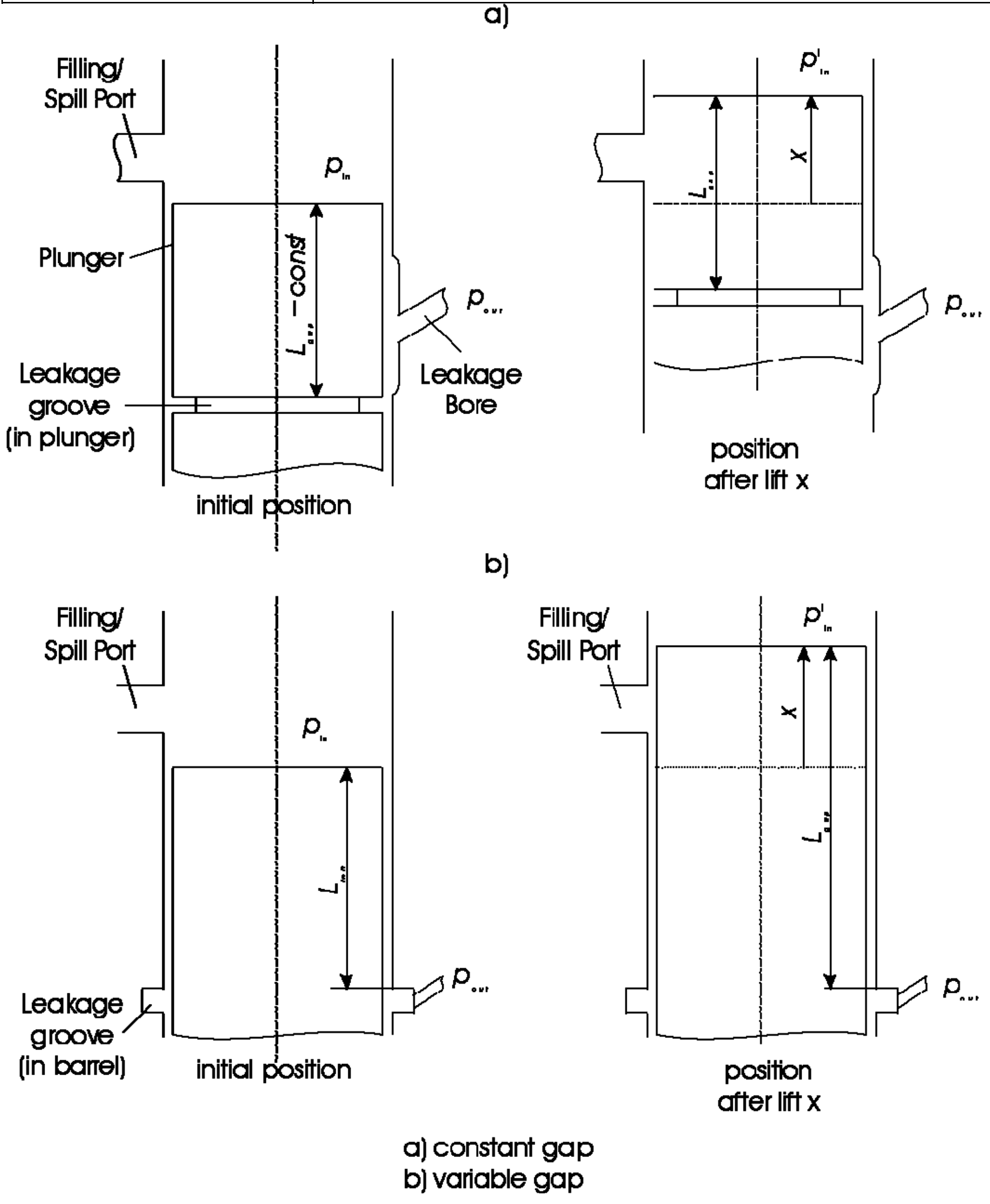


Figure 110: Gap Length for a Plunger of a Mechanical Pump

Program-calculated annular gap (compliant volume)	Click to use program-calculated annular gap. **Note: Annular Gap** element must have at least one hydraulic connection to **Compliant Volume** element and, furthermore, in the input dialog of this **Compliant Volume** the toggle button 'Cylindrical Shape' and check box 'Compliant Walls' are switched on. **Note:** If there is connection to **Compliant Volume** element, the "Program-calculated annular gap" option cannot be activated (faded out).
User-defined annular gap (standard volume)	Specify values of annular gap (clearance) as a function of pressure (average pressure between input and output is implied here) (see Chapter 13.1.5.2). If diametral gap is constant (invariant of pressure), only one value has to be specified in the table. In this case no pressure value is needed (1st column empty). Diametral gap is defined by: $\Delta D = D_b - D_p$ where D_b and D_p are diameters of barrel and piston, respectively.
User-defined Variable Gap (3D Table)	
Diametral Gap vs. Pressure and Distance	Press this bar to specify the diametral gap D_i as a function of pressure difference Δp and distance from input side of Plunger/Piston L_i (at the start of annular gap) in the form of 3D table (see Chapter 13.1.5.3). Where pressure difference Δp is difference between pressures on input and output end of annular gap. $\Delta p = \lvert p_{inp} - p_{out} \rvert$

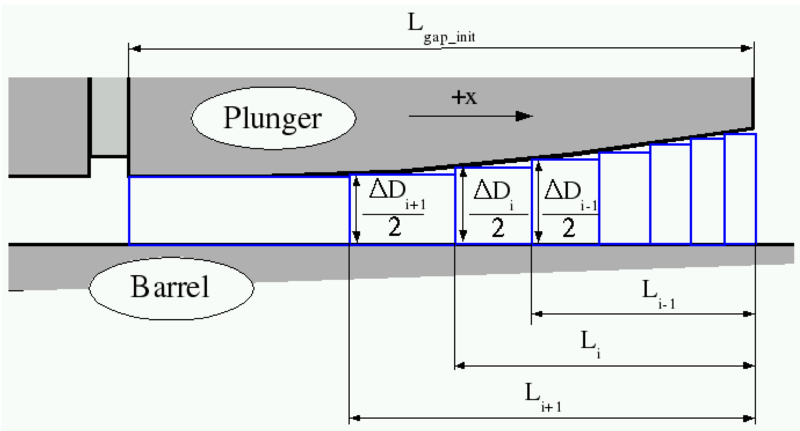


Figure 111: Geometry of User Defined Variable Gap (constant gap length)

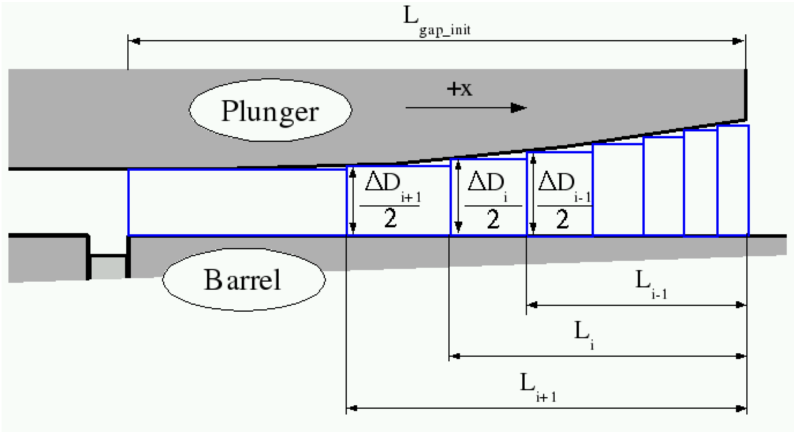


Figure 112: Geometry of User Defined Variable Gap (variable gap length)

13.1.2. Initial Conditions

Initial conditions cannot be specified for a **Leakage | Annular Gap** element.

13.1.3. Modify Parameter

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

inner gap (piston) diameter	SI Unit:	m
length of annular gap	SI Unit:	m
diametral clearance of gap (to be used only with pressure-invariant annular gap)	SI Unit:	m

13.1.4. Output Parameters

Path: **Element** | **Store Results**

The **Output Parameters Dialog Box** of **Annular Gap** is shown in Figure 113.

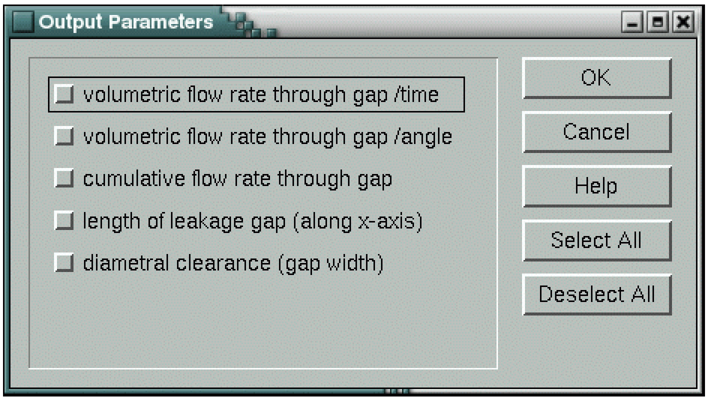


Figure 113: Output Parameters of Annular Gap Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

13.1.5. Leakage Model

The **Leakage** model is based on the Hagen-Poiseuille law[26]. It considers steady laminar flow through **Annular Gap** because small cross-sectional gap area results in laminar flow. As fluid enters the **Annular Gap,** the velocity profile is linear. The fluid velocity at the barrel wall is equal to barrel velocity and at the piston wall is equal to piston velocity. This layer of fluid exerts considerable shear forces on the inner layers whose velocities must exceed piston velocity v_p to satisfy the law of continuity.

The boundary layers (thus formed) increase in thickness until the center of the **Annular Gap** is reached. The velocity profile then becomes parabolic. It remains parabolic throughout the length of **Annular Gap** (refer to Figure 114**).**

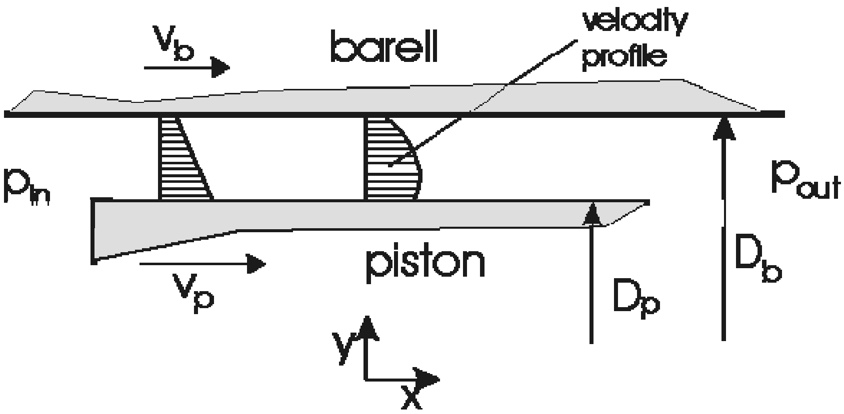


Figure 114: Laminar Flow through Annular Gap

The following notations are used in Figure 114:

R_p - radius of piston

R_b - inner radius of the barrel

v_p - velocity of piston

v_b - velocity of barrel

[26] Bohl, W. Technische Strömungslehre. 7-e Auflage. Würzburg, Vogel-Buchverlag, 1986.

p_{in} - pressure on input side of piston

p_{out} - pressure on output side of piston

For steady laminar flow through **Annular Gap**, Navier-Stokes Equation (**152**) is reduced to:

$$\frac{dp}{dx} = \mu \frac{d^2 v}{dy^2} \tag{227}$$

where:

x,y - coordinates of motion

v - velocity of fluid in x direction

p - pressure

μ - dynamic viscosity ($\mu = \nu\rho$ where ν is kinematic viscosity).

Integrating Equation 227 twice and rearranging terms, we obtain fluid velocity profile as function of y:

$$v(y) = \frac{1}{\mu}\frac{dp}{dx}\frac{y^2}{2} + C_1 y + C_2 . \tag{228}$$

Constants C_1 and C_2 can be calculated from boundary conditions given by:

$$\begin{aligned} v &= v_p, \quad y = R_p, \\ v &= v_b, \quad y = R_b. \end{aligned} \tag{229}$$

Substituting these boundary conditions into Equation 228, we obtain:

$$v(y) = T_1 y^2 + \left\{ \frac{v_b - v_p}{R_b - R_p} - T_1\left(R_b + R_p\right) \right\} y + v_b - \frac{v_b - v_p}{R_b - R_p} R_b + T_1 R_b R_p , \tag{230}$$

where $T_1 = \frac{1}{2\mu}\frac{dp}{dx}$.

Furthermore we assume linear pressure gradient along leakage path (piston length):

$$\frac{dp}{dx} = \frac{p_{out} - p_{in}}{L_{gap}} , \tag{231}$$

where L_{gap} is the actual gap length.

Flow rate through leakage gap is defined by:

$$\dot{Q} = \int_{R_p}^{R_b} v(y) 2\pi y dy . \tag{232}$$

Inserting Equations 230 and 231 into Equation 232 and performing integration, we obtain the formula for leakage flow rate:

$$\dot{Q} = \frac{\pi}{\mu}\frac{p_{out} - p_{in}}{L_{gap}}\left[\frac{1}{4}\left(R_b^4 - R_p^4\right) - \frac{1}{3}\left(R_b + R_p\right)\left(R_b^3 - R_p^3\right) + \frac{1}{2}R_b R_p\left(R_b^2 - R_p^2\right)\right] +$$

$$+\pi\left(v_b - v_p\right)\left[\frac{2}{3}\frac{R_b^3 - R_p^3}{R_b - R_p} - R_p\left(R_b + R_p\right)\right] + \pi v_b\left(R_b^2 - R_p^2\right). \quad (233)$$

13.1.5.1. Calculation of Shear Force

Viscous shear forces acting on piston and barrel can be defined as:

$$F_{shear\_p} = \tau_p 2\pi R_p L_{gap},$$

$$F_{shear\_b} = \tau_b 2\pi R_p L_{gap}, \quad (234)$$

where τ is shear stress given by:

$$\tau = \mu\frac{dv}{dy} = \mu\left\{2T_1 y + \left(\frac{v_b - v_p}{R_b - R_p} - T_1\left(R_b + R_p\right)\right)\right\}. \quad (235)$$

Clearly, for piston boundary layer $y=R_p$ and for barrel boundary layer $y=R_b$.

Substituting Equation 235 into Equation 234, we arrive to the following expression for viscous friction force acting on piston:

$$F_{shear\_p} = \pi R_p\left(p_{in} - p_{out}\right)\left(R_p - R_b\right) + 2\pi\mu\frac{R_p}{R_b - R_p}L_{gap}\left(v_b - v_p\right),$$

$$F_{shear\_b} = \pi R_b\left(p_{in} - p_{out}\right)\left(R_b - R_p\right) + 2\pi\mu\frac{R_b}{R_b - R_p}L_{gap}\left(v_b - v_p\right). \quad (236)$$

If a mechanical body (**Piston**, **Plunger**, **Needle** guide, etc.) is connected by special connection to a **Leakage** element, viscous friction force will be automatically added to its equation of motion.

13.1.5.2. Calculation of variable clearance

If the dependence of annular gap width on pressure has to be encountered and no external data are available, the radial displacement of barrel wall can be calculated from the equation:

$$u(r) = \frac{\left(1 - \eta_b\right)\left(p_b R_b^2 - p_{bo}R_{bo}^2\right)r + \left(1 + \eta_b\right)\left(p_b - p_{bo}\right)\frac{R_b^2 R_{bo}^2}{r}}{E_b\left(R_{bo}^2 - R_b^2\right)}, \quad (237)$$

where:

r – distance of wall point from the barrel axis

R_b – inner radius of the barrel (defined previously)

R_{bo} – outer radius of the barrel

p_b – pressure inside the barrel (pump pressure, usually $p_b = p_{out}$)

p_{bo} – pressure outside the barrel (leak pressure, usually $p_{bo} = p_{in}$)

E_b – Young's modulus of barrel wall material (for steel E_b=210000 N/mm^2)

η_b – Poisson's ratio of barrel wall material (for steel ν_b=0.3)

Equation 237 is valid for the homogeneous thick-walled pressure vessel of cylindrical shape. For the calculation of annular gap width, only the point $r=R_b$ is of interest. Then the diametral gap width is given by:

$$u(D_b) = 2u(R_b) = \frac{2R_b\left\{(1-\eta_b)\left(p_b R_b^2 - p_{bo}R_{bo}^2\right) + (1+\eta_b)(p_b - p_{bo})R_{bo}^2\right\}}{E_b\left(R_{bo}^2 - R_b^2\right)} \quad . \qquad (238)$$

Note that in the table of input dialog of **Leakage | Annular Gap** element the diametral gap has to be specified as a function of mean pressure between input and output:

$$u(D_b) = f\left(\frac{|p_{out} - p_{in}|}{2}\right)$$

13.1.5.3. Calculation of Flow Rate for User-defined Variable Gap

Flow rate through each section of annular gap (refer to Figure 111 and Figure 112) can be calculated from the equation:

$$\dot{Q}_i = \frac{\pi}{\mu}\frac{\Delta p_i}{\Delta L_i}\left[\frac{1}{4}\left(R_{bi}^4 - R_{pi}^4\right) - \frac{1}{3}\left(R_{bi} + R_{pi}\right)\left(R_{bi}^3 - R_{pi}^3\right) + \frac{1}{2}R_{bi}R_{pi}\left(R_{bi}^2 - R_{pi}^2\right)\right] +$$

$$+\pi\left(v_b - v_p\right)\left[\frac{2}{3}\frac{R_{bi}^3 - R_{pi}^3}{R_{bi} - R_{pi}} - R_{pi}\left(R_{bi} + R_{pi}\right)\right] + \pi v_b\left(R_{bi}^2 - R_{pi}^2\right).$$

If it is assumed that

$$\dot{Q}_1 = \dot{Q}_2 = ... = \dot{Q}_i = \dot{Q}$$

and

$$p_{out} - p_{in} = \sum_i \Delta p_i$$

we obtain the formula for leakage flow rate:

$$\dot{Q} = \frac{p_{out} - p_{inp}}{\sum_i \frac{1}{S1}} \qquad (239)$$

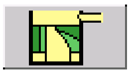
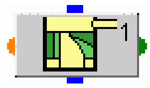
where:

$$S1 = \frac{\pi}{\mu}\frac{1}{\Delta L_i}\left[\frac{1}{4}\left(R_{bi}^4 - R_{pi}^4\right) - \frac{1}{3}\left(R_{bi} + R_{pi}\right)\left(R_{bi}^3 - R_{pi}^3\right) + \frac{1}{2}R_{bi}R_{pi}\left(R_{bi}^2 - R_{pi}^2\right)\right] +$$

$$+\pi\left(v_b - v_p\right)\left[\frac{2}{3}\frac{R_{bi}^3 - R_{pi}^3}{R_{bi} - R_{pi}} - R_{pi}\left(R_{bi} + R_{pi}\right)\right] + \pi v_b\left(R_{bi}^2 - R_{pi}^2\right).$$

14. PORT

14.1. In-line Fill/Spill Port

Element Name:	**In-line Fill/Spill Port**	
Element Icon:		
Definition:	This element serves to define a round filling, spill, or fill/spill port of in-line pump.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connection can be specified.	

☞ **Note:** In the In-line pump model, round **Fill/Spill Port** must be used in combination with **Plunger** element (via special connection). Information about plunger velocity will be transferred via special connection.

The schematic of the **In-line Spill Port** with **Plunger** geometry (upper and lower helix) is shown in Figure 115.

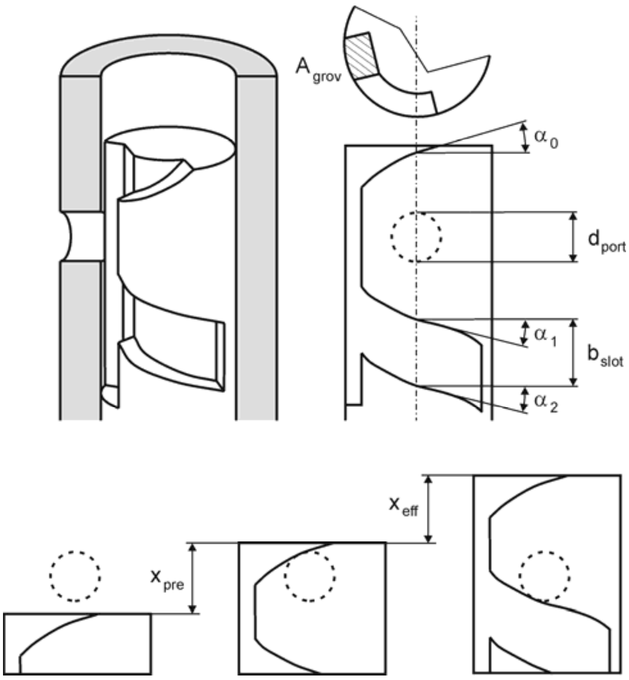
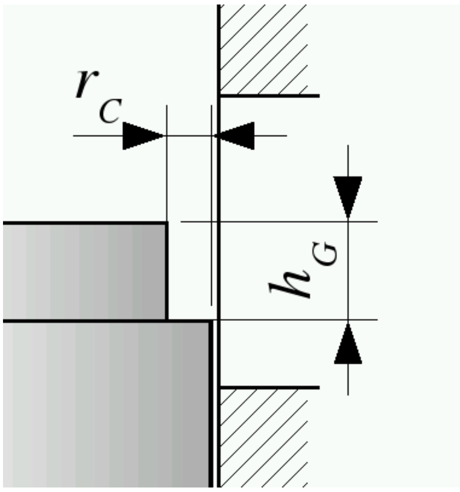
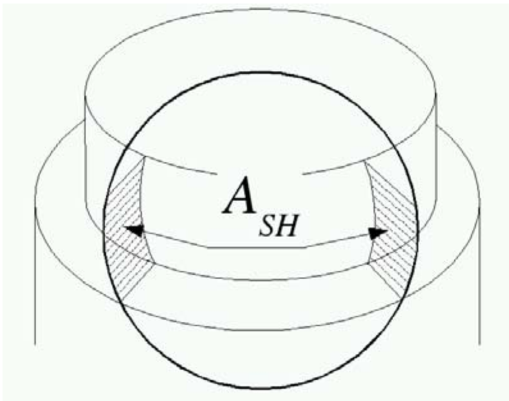


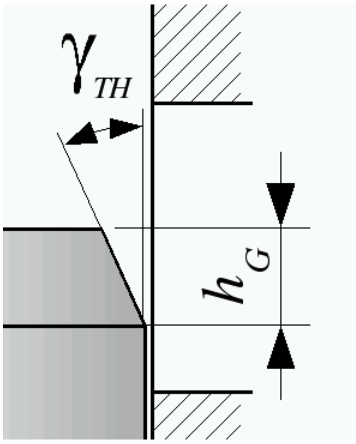
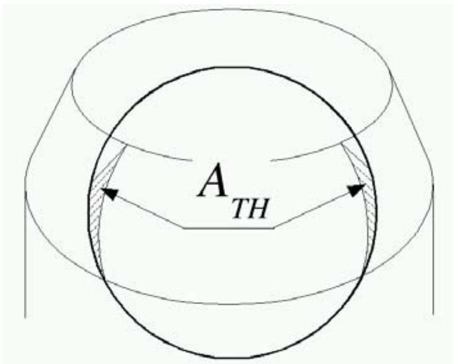
Figure 115: In-line Fill/Spill Port and Plunger Geometry

14.1.1. Input Parameters

Description of input data for the **Fill/Spill Port**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
PLUNGER DATA	
Fill helix angle	Specify upper (fill) helix angle α_0. **Note:** Fill helix angle is automatically disabled if Special Head Geometry is active.
Stroke: Pre-stroke/ Effective stroke	Specify **Pre-stroke** x_{pre} (roller on cam base circle) and **Effective stroke** x_{eff} (position of plunger at port opening). **Pre-stroke** is a distance between plunger top and upper fill/spill port edge at the plunger position when its roller is located on cam base circle (bottom position) (refer to Figure 115). In other words, **Pre-stroke** is the stroke plunger makes from its bottom position to the point where the fill/spill port is just covered, as the plunger lifts. **Effective stroke** of plunger is defined as distance between plunger top (output end) and upper spill port edge at the plunger position when its metering ramp reaches the lower edge of spill port (refer to Figure 115). In other words, **Effective stroke** is the lift plunger travels from the beginning of fill/spill port cutoff (after **Pre-stroke**) until the spill port just starts to be uncovered by the rising plunger. **Note:** Effective stroke is a dynamic effective stroke. It is compared during each calculation step with the actual plunger lift after pre-stroke (refer to Section 14.1.5). Hence effective stroke has to include the plunger-tappet deformation, shaft bending and torsion etc. (if relevant).
Helix angle: First/Second	Specify **First-** and **Second** helix angle (α_1 and α_2) of metering ramp (refer to Figure 115). Slot width is the distance the plunger makes from the position where spill port starts to be uncovered (after **Effective stroke**) to the position when it starts to be covered again (or max. plunger position, if spill port is not covered anymore).
Plunger slot width	Specify plunger slot width b_{slot}.
Plunger diameter	Specify plunger diameter. If it is not specified, it will be taken from element that is connected to Fill/Spill Port via special connection.
Cross-section of plunger groove/bore: Narrowest area	Specify narrowest cross-section area of plunger groove or central bore A_{grov} (depending on the plunger type).
Special Head Geometry	Press this button to define one of three possible special head geometries.
Shouldered/ tapered	

Head Groove Data …	Press this bar to specify geometry of shouldered or tapered plunger head.
Shouldered Head	 **Figure 116: Shouldered Head Geometry**
Groove depth	Specify groove depth h_G.
Radial clearance	Specify radial clearance r_C.
Flow discharge coefficient	Specify flow discharge coefficient μ_{SH}.  **Figure 117: Shouldered Head Flow Area** Flow rate through head groove area into port for shouldered head is given by: $$\dot{Q}_{HG} = \dot{Q}_{SH} = sign(\Delta p)\mu_{SH} A_{SH} \sqrt{\frac{2\lvert\Delta p\rvert}{\rho}}$$

Tapered Head	 **Figure 118: Tapered Head Geometry**
Groove depth	Specify groove depth h_G.
Taper angle	Specify taper angle γ_{TH}.
Flow discharge coefficient	Specify flow discharge coefficient μ_{TH}.  **Figure 119: Tapered Head Flow Area** Flow rate through head groove area into port for tapered head is given by: $$\dot{Q}_{HG} = \dot{Q}_{TH} = sign(\Delta p)\mu_{TH} A_{TH} \sqrt{\frac{2\lvert\Delta p\rvert}{\rho}}$$
Relief grinding	
Head Grinding Data …	Press this bar to specify geometry of plunger head relief grinding.

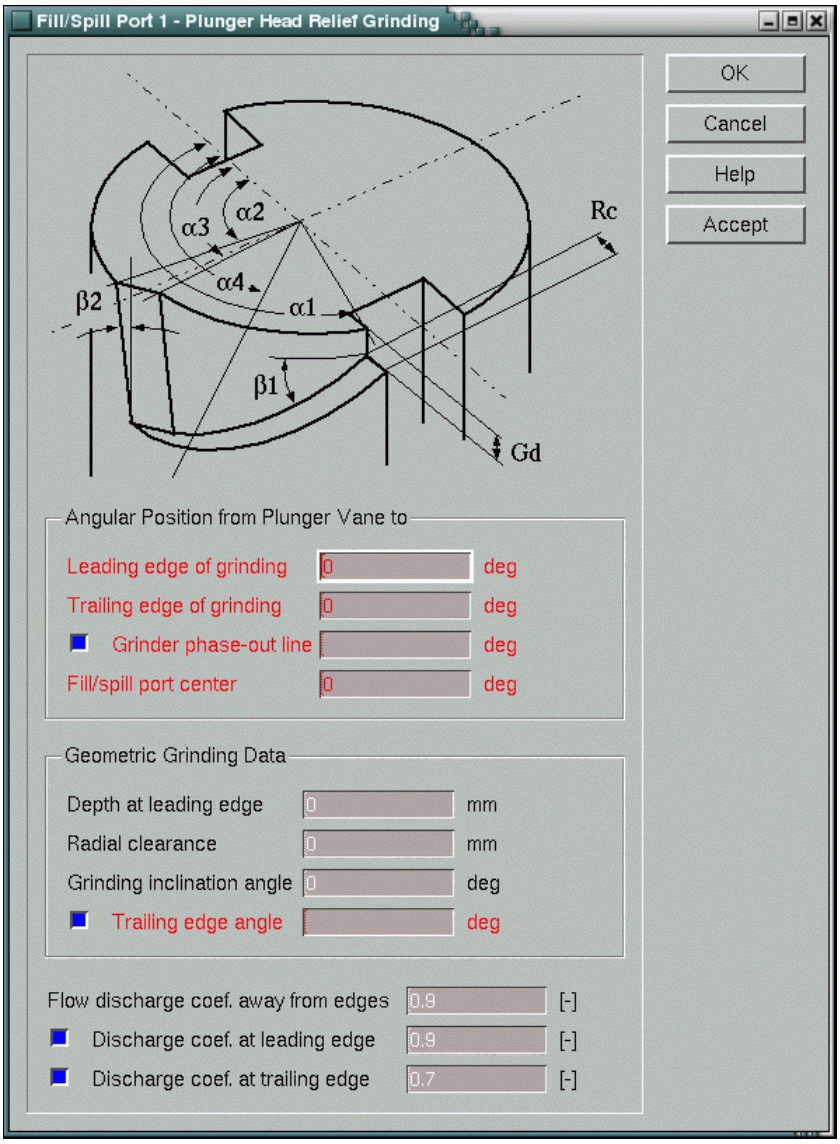
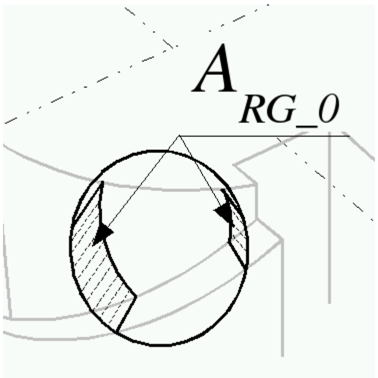
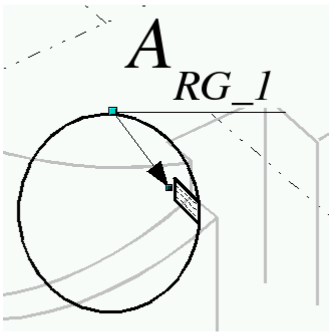
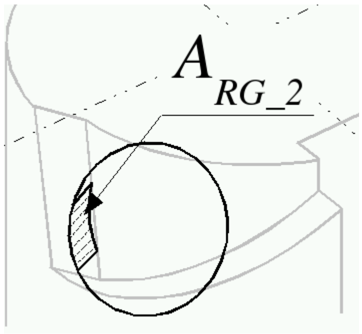


Figure 120: Plunger Head Relief Grinding Dialog

Angular position from Plunger Vane to	
Leading edge of grinding	Specify angle between plunger vane and leading edge of grinding α_1.
Trailing edge of grinding	Specify angle between plunger vane and trailing edge of grinding α_2.
Grinder phase-out line	Specify angle between plunger vane and grinder phase-out line α_3.
Fill/spill port center	Specify angle between plunger vane and fill/spill port centerline α_4.
Geometric Grinding Data	
Depth at leading edge	Specify depth at leading edge G_d.
Radial clearance	Specify radial clearance of relief grinding R_C.

Grinding inclination angle	Specify inclination angle of relief grinding β_1.
Trailing edge angle	Specify trailing edge angle β_2. **Note:** Trailing edge angle is not implemented.
Flow discharge coefficient away from edges	Specify flow discharge coefficient away from edges μ_{RG_0}.  **Figure 121: Head Relief Grinding Flow Area** Flow rate through head groove into port when port is away from edges is given by: $\dot{Q}_{RG\_0} = sign(\Delta p)\mu_{RG\_0}A_{RG\_0} = \sqrt{\frac{2\lvert\Delta p\rvert}{\rho}}$
Flow discharge coefficient at leading edge	Specify flow discharge coefficient at leading edge μ_{RG_1}.  **Figure 122: Flow area near to leading edge** Flow rate through part of head groove into port near to leading edge is given by: $\dot{Q}_{RG\_1} = sign(\Delta p)\mu_{RG\_1}A_{RG\_1} = \sqrt{\frac{2\lvert\Delta p\rvert}{\rho}}$

Flow discharge coefficient at trailing edge	Specify flow discharge coefficient at trailing edge μ_{RG_2}.  **Figure 123: Flow area near to trailing edge** Flow rate through part of head groove into port at grinder phase-out area is given by: $\dot{Q}_{RG\_2} = sign(\Delta p)\mu_{RG\_2} A_{RG\_2} = \sqrt{\frac{2\lvert\Delta p\rvert}{\rho}}$ Flow rate through head groove into port for head relief grinding is sum of all three flow rates: $\dot{Q}_{HG} = \dot{Q}_{RG\_0} + \dot{Q}_{RG\_1} + \dot{Q}_{RG\_2}$ **Note:** Flow rate through head groove $\dot{Q}_{HG}$ is added to flow rates through lower helix (see Chapter 14.1.5) and flow rate directly from pump chamber to port.
Port diameter	Specify diameter of the port d_{port} (refer to Figure 126).
Number of ports	Specify number of ports. If there are two or more ports with equal geometries, they can be defined within one **Fill/Spill Port** element. The ports must be identical.
Program-calculated flow resistance of port	Click to use internal flow resistance function $\xi_{port} = f\,(A_{min} / A_{spill})$ where A_{min} is the narrowest open cross-sectional flow area and A_{sill} is the largest cross-sectional area of spill port (refer to Figure 124). Flow resistance of port ξ is calculated according to formulas in Chapter 14.1.5.

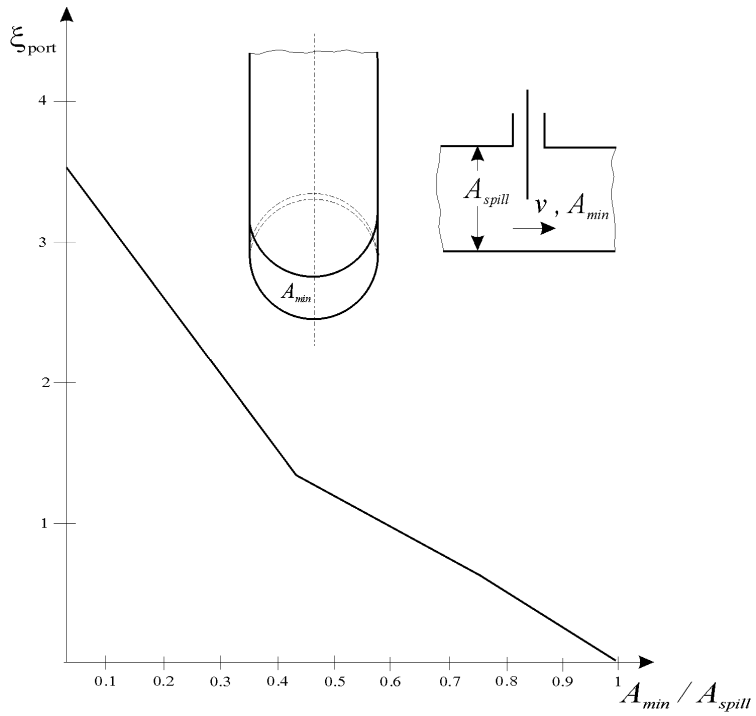


Figure 124: Flow Resistance of Fill/Spill Port (gate valve in a straight tube)

User-defined flow resistance of port	Click to activate the table for flow resistance coefficient as a function of the narrowest open cross-sectional flow area $\xi = f(A_{min})$. Enter values of open cross-sectional area A_{min} of fill/spill port in first column and corresponding values of flow resistance coefficient ξ in second column. The specified values are bridged by linear interpolation. The values of cross-sectional area A_{min} must be arranged in ascending order. If the actual cross-sectional area is less than the area specified in the first table row, the value of resistance coefficient from the first row will be used (no extrapolation performed). Analogously, if the actual cross-sectional area exceeds the area in the last table row (maximum area), the value of resistance coefficient from the last row will be used for further calculation (until the area gets back into the defined range again). If flow resistance in fill/spill port is constant (invariant of cross-sectional area), thus only one value has to be specified in the 2nd column of the table. In this case no cross-sectional area value is needed (1st column empty).

☞ **Note:** Open cross-sectional area *A* of **Fill/Spill Port** is the minimum flow area in flow path from pump volume via lower helix to spill port outlet. The area is either a spill port area at actual plunger position or cross-section area of the plunger groove/bore.

14.1.2. Initial Conditions

Initial conditions cannot be specified for **In-line Fill/Spill Port** element.

14.1.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 22.2.4.1). Open by pressing **Modify...** bar.

port diameter	SI Unit:	m
flow resistance coefficient of port	SI Unit:	---
pre-stroke of plunger	SI Unit:	m
effective stroke of plunger	SI Unit:	m
narrowest area of plunger groove	SI Unit:	m^2

14.1.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameters Dialog Box of In-line Fill/Spill Port** is shown in Figure 125.

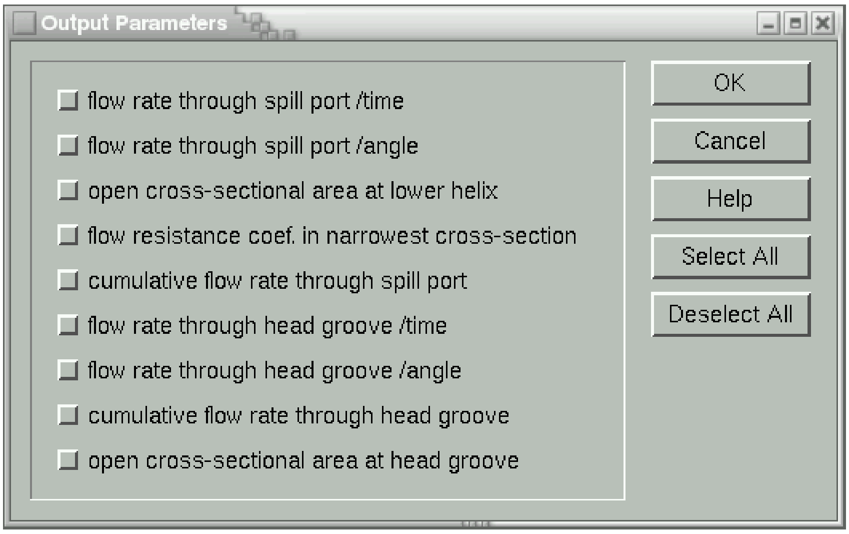


Figure 125: Output Parameters of In-line Round Fill/Spill Port Dialog

To activate output parameter, the check box button in front of output parameter has to be checked.

14.1.5. Flow Equation

BOOST Hydsim model of **In-Line Fill/Spill Port** is based on the flow through a gate valve in a straight tube (refer to Figure 126).

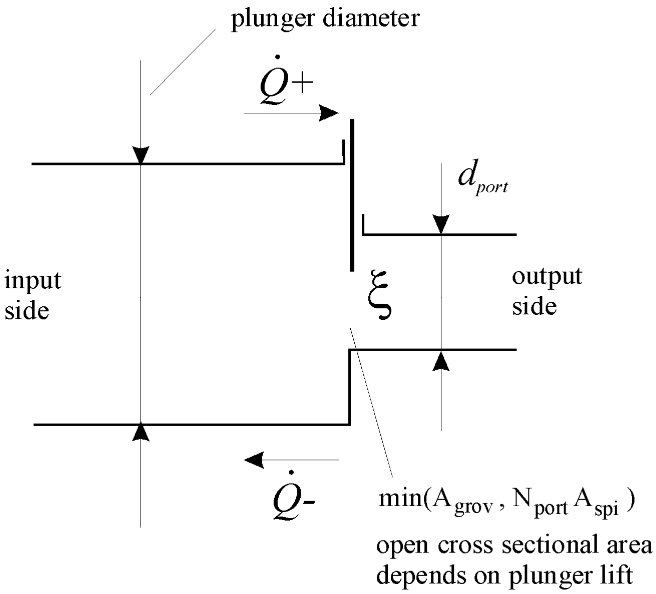


Figure 126: Flow Model through Fill/Spill Port

Flow rate through a single spill port is calculated from Bernoulli equation:

$$\dot{Q}_i = sign(\Delta p) A_{\min} \sqrt{\frac{1}{\left(\frac{A_{\min}}{A_{spill}}\right)^2 - \left(\frac{A_{\min}}{A_{plung}}\right)^2 + \xi} \cdot \frac{2|\Delta p|}{\rho}} \tag{240}$$

with:

$\Delta p = \lvert p_{in} - p_{out} \rvert$	pressure difference between input and output
A_{min}	minimal flow area (cross-section area of plunger groove/bore or calculated spill port area). Refer to Equation 242.
$A_{plung} = \frac{\pi}{4} d_{plung}^2$	Cross-sectional area of plunger (input end)
$A_{spill} = \frac{\pi}{4} d_{port}^2$	Cross sectional area of spill port (output end)
ξ	Flow resistance of port

User-defined flow resistance of port	Program calculated flow resistance of port	
$\xi = f(A_{min})$	$\Delta p > 0$	$\Delta p < 0$
	$\xi = \frac{1}{2}\left(\frac{A_{min}}{A_{spill}}\right)^2 + \xi_{port}$	$\xi = \left(1 - \frac{A_{spill}}{A_{plung}}\right)^2 \left(\frac{A_{min}}{A_{spill}}\right)^2 + \xi_{port}$

Total flow rate is given by:

$$\dot{Q} = N_{port} \cdot \left(\dot{Q}_i + \dot{Q}_{HG}\right) \tag{241}$$

where N_{port} is number of identical ports.

Open cross-sectional area of port is minimum between cross-section area of groove A_{grov} and total cross-sectional area of all spill ports:

$$A_{min} = min\left(N_{port} A_{spill}, A_{grov}\right) \tag{242}$$

The definition of flow areas A_{grov} and A_{spill} is shown in Figure 127.

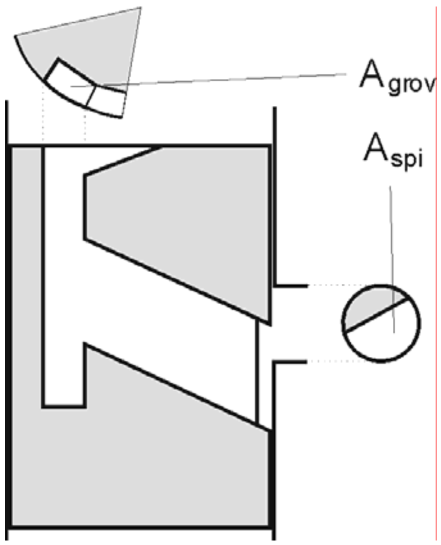
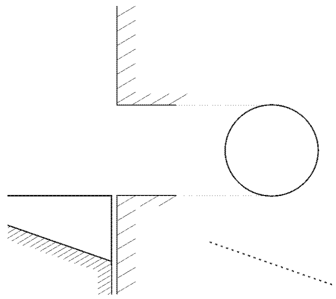


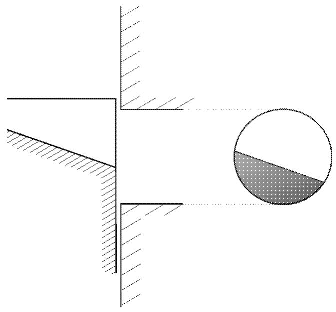
Figure 127: Cross-sectional Flow Area of Fill/Spill Port and Plunger Groove

Table7: Cross-sectional Flow Areas through Fill/Spill Port

UPPER HELIX ANGLE



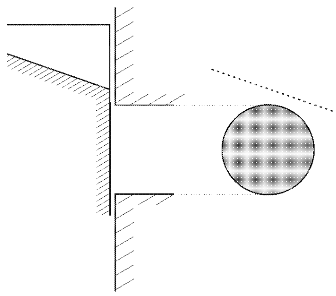
$$x_{plunger} \le x_{pre} - \frac{2R_{port}}{\cos\alpha_0} \qquad A_{QDR1} = R_{port}^2 \pi \tag{243}$$



$$h = 2R_{port} - \left(x_{plunger} - x_{pre} + \frac{2R_{port}}{\cos\alpha_0} \right) \cos\alpha_0 \tag{244}$$

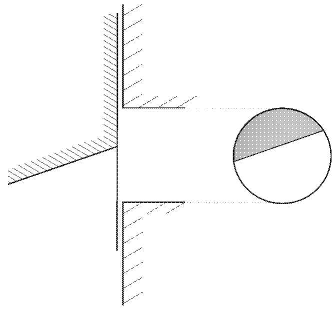
$$x_{pre} - \frac{2R_{port}}{\cos\alpha_0} < x_{plunger} < x_{pre} \qquad \widehat{\varphi} = 2\arccos\left(\frac{R_{port} - h}{R_{port}} \right)$$

$$A_{QDR1} = \frac{1}{2} R_{port}^2 \left(\widehat{\varphi} - \sin\widehat{\varphi} \right) \tag{245}$$

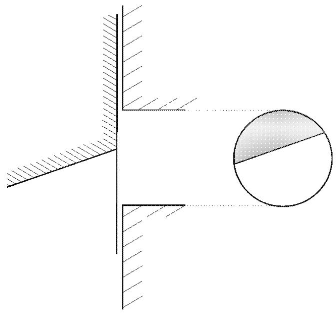


$$x_{plunger} \ge x_{pre} \qquad A_{QDR1} = 0 \tag{246}$$

RST LOWER HELIX ANGLE



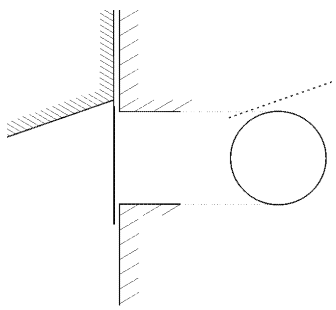
$$x_{plunger} \le x_{pre} + x_{eff} \qquad A_{QDR2} = 0 \tag{247}$$



$$h = \left(x_{plunger} - x_{pre} - x_{eff}\right)\cos\alpha_1 \tag{248}$$

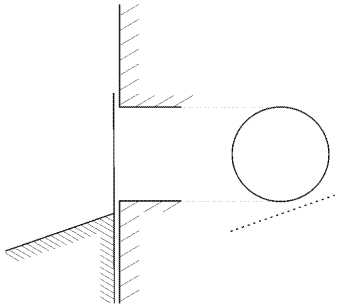
$$x_{pre} + x_{eff} < x_{plunger} < x_{pre} + x_{ef} + \frac{2R_{port}}{\cos\alpha_1} \qquad \widehat{\varphi} = 2\arccos\left(\frac{R_{port} - h}{R_{port}}\right)$$

$$A_{QDR2} = \frac{1}{2} R_{port}^2 \left(\widehat{\varphi} - \sin\widehat{\varphi}\right) \tag{249}$$

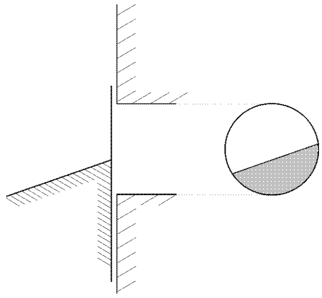


$$x_{plunger} \ge x_{pre} + x_{ef} + \frac{2R_{port}}{\cos\alpha_1} \qquad A_{QDR2} = R_{port}^2 \pi \tag{250}$$

SECOND LOWER HELIX ANGLE



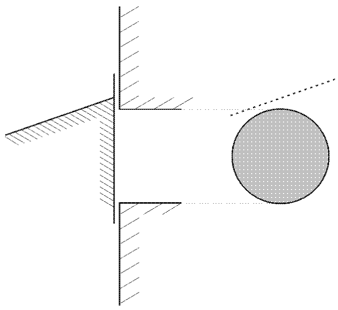
$$x_{plunger} \le x_{pre} + x_{eff} + b_{slot} \qquad A_{QDR3} = 0 \tag{251}$$



$$h = \left(x_{plunger} - x_{pre} - x_{eff} - b_{slot}\right)\cos\alpha_2 \tag{252}$$

$$x_{pre} + x_{eff} + b_{slot} < x_{plunger} < x_{pre} + x_{eff} + b_{slot} + \frac{2R_{port}}{\cos\alpha_2} \qquad \hat{\varphi} = 2\arccos\left(\frac{R_{port} - h}{R_{port}}\right)$$

$$A_{QDR3} = \frac{1}{2} R_{port}^2 \left(\hat{\varphi} - \sin\hat{\varphi}\right) \tag{253}$$



$$x_{plunger} \ge x_{pre} + x_{eff} + b_{slot} + \frac{2R_{port}}{\cos\alpha_2} \qquad A_{QDR3} = R_{port}^2 \pi \tag{254}$$

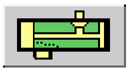
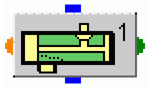
ACTUAL CROSS-SECT. AREA OF SPILL PORT IS:

$$A_{spi} = A_{QDR1} + A_{QDR2} - A_{QDR3} \tag{255}$$

or if $\quad A_{groov} > 0$ and $x_{plunger} > x_{pre}$ and $n_{ports} A_{QDR} > A_{groov}$

$$A_{spi} = A_{groov} \tag{256}$$

14.2. Distributor Fill/Spill Port

Element Name:	**Distributor Filling and Spill Ports**	
Element Icon:		
Definition:	This element serves to define the filling/spill ports of distributor-type rotary pump (Bosch VP etc.).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connections must be specified.	

☞ **Note: Fill/Spill Ports of Distributor Pump** must be exclusively used with **Cam Plate** element (not **Cam Profile**) and attached **Plunger** element (via special connection).

The schematic of **Distributor Pump with Filling and Spill Ports** is shown in Figure 128 (front view) and Figure 129 (top view).

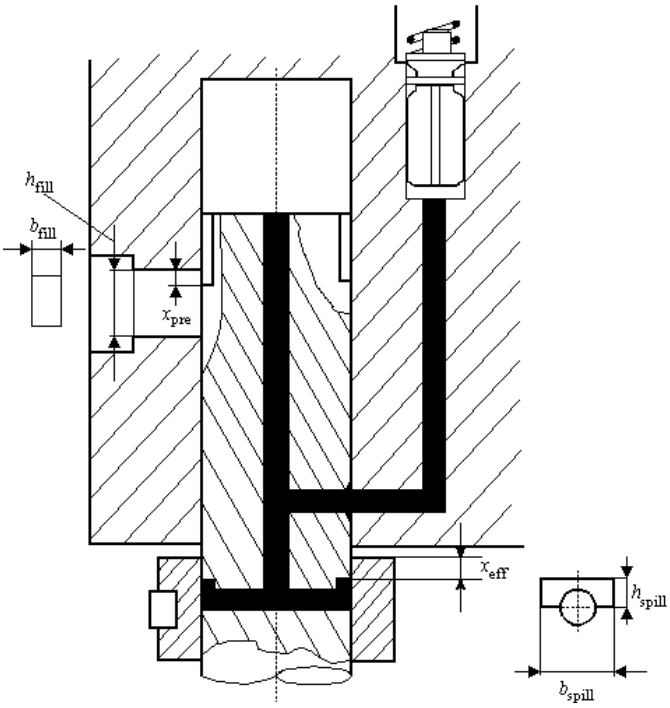


Figure 128: Distributor Pump with Filling and Spill Ports (front view)

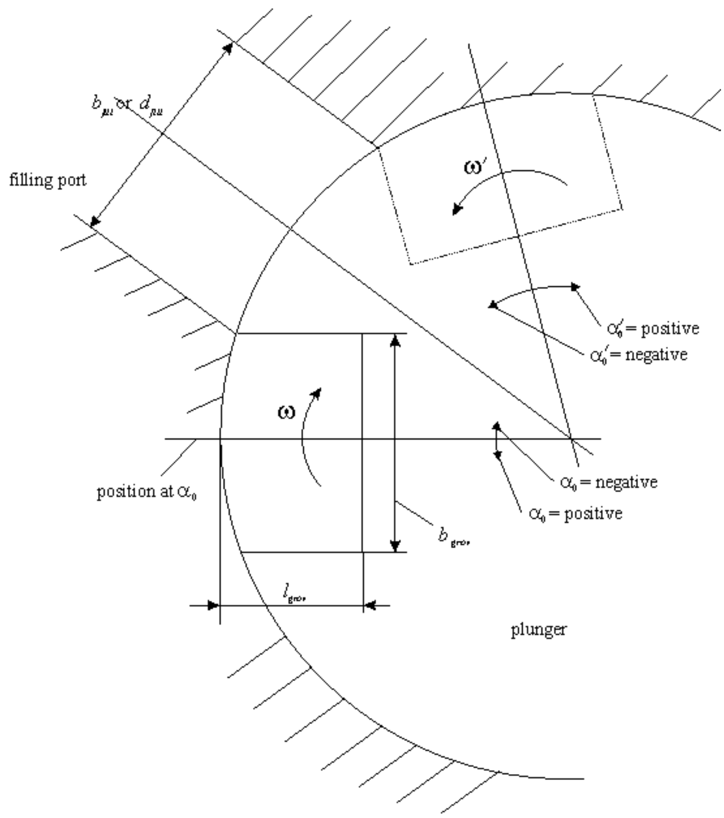


Figure 129: Distributor Pump with Filling and Spill Ports (top view)

14.2.1. Input Parameters

Description of input data for the **Filling and Spill Ports**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
FILLING PORT	
Rectangular shape	Click to specify rectangular shape of the filling port.
Port width size	Specify width b_{fill} of the filling port (refer to Figure 128).
Port height size	Specify height h_{fill} of the filling port (refer to Figure 128).
Circular shape	Click to specify circular shape of the filling port.
Port diameter	Specify diameter of the filling port (refer to Figure 128).
Program-calculated flow resistance of port	Click to use internal flow resistance function $\xi_{fill} = f(A)$ (gate valve in a straight tube) (refer to Figure 124).
User-defined flow resistance	Click to activate the table for flow resistance as a function of open cross-sectional fill area (function. $\xi_{fill} = f(A)$) Enter values of open cross-sectional area A_{fill} of filling port in first column and values of flow resistance coefficient ξ_{fill} in second column. The specified values are bridged by linear interpolation. Refer to the description of **Spill Port** flow resistance coefficient table for more information. If flow resistance in spill port is constant (invariant of cross-sectional area), only one value has to be specified in the 2nd column of the table. In this case, no cross-sectional area value is needed (1st column empty).
PLUNGER GROOVE	
Width	Specify width of groove b_{grov} (refer to Figure 129).
Depth	Specify depth of plunger groove in l_{grov} (refer to Figure 129).
Initial angle to filling port	Specify angle of orientation of plunger-groove to filling port at start of calculation α_0 (refer to Figure 129).
PLUNGER	
Stroke: Pre-stroke/ Effective Stroke	Specify **Pre-stroke** X_{pre} (roller on cam base circle) and **Effective stroke** X_{eff} (position of plunger at port opening). Refer to Figure 128.
Plunger diameter	Specify plunger diameter. If it is not specified, it will be taken from element connected to Distributor Fill/Spill Port via special connection.
SPILL PORT(S)	
Number of spill ports	Specify number of spill ports.
Rectangular shape	Click to specify rectangular shape of the spilling port.
Port width size	Specify width b_{spill} of the spilling port (refer to Figure 128).

Port height size	Specify height h_{spill} of the spilling port (refer to Figure 128).
Circular shape	Click to specify circular shape of the spilling port.
Port diameter	Specify diameter of the spilling port (refer to Figure 128).
Program-calculated flow resistance of port	Click to use internal flow resistance function of spill port $\xi_{spill} = f(A/A_0)$ (gate valve in a straight tube) (refer to Figure 124).
User-defined flow resistance	Click to activate the table for flow resistance as a function of narrowest open cross-sectional area function $\xi_{spill} = f(A)$ Enter values of open cross-sectional area A_{spill} of spill port in first column and values of flow resistance coefficient ξ_{spill} in second column. The specified values are bridged by linear interpolation. The values of cross-sectional area *A* must be arranged in ascending order. If the actual cross-sectional area is less than the area specified in the first table row, the value of resistance coefficient from the first row will be used (no extrapolation performed). Analogously, if the actual cross-sectional area exceeds the area in the last table row (maximum area), the value of resistance coefficient from the last row will be used for further calculation (until the area gets back into the defined range again). If flow resistance in spill port is constant (invariant of cross-sectional area), only one value has to be specified in the 2nd column of the table. In this case no cross-sectional area value is needed (1st column empty).

14.2.2. Initial Conditions

Initial conditions cannot be specified for **Distributor Filling and Spill Ports**.

14.2.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

width of rectangular spill port	SI Unit:	m
height of rectangular spill port	SI Unit:	m
flow resistance coefficient of spill port	SI Unit:	---
effective stroke of plunger	SI Unit:	m
flow resistance coefficient of filling port	SI Unit:	---

width of rectangular filling port	SI Unit:	m
height of rectangular filling port	SI Unit:	m
diameter of circular spill port	SI Unit:	m
diameter of circular filling port	SI Unit:	m

14.2.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameters Dialog Box** of **Distributor Filling and Spill Ports** element is shown in Figure 130.

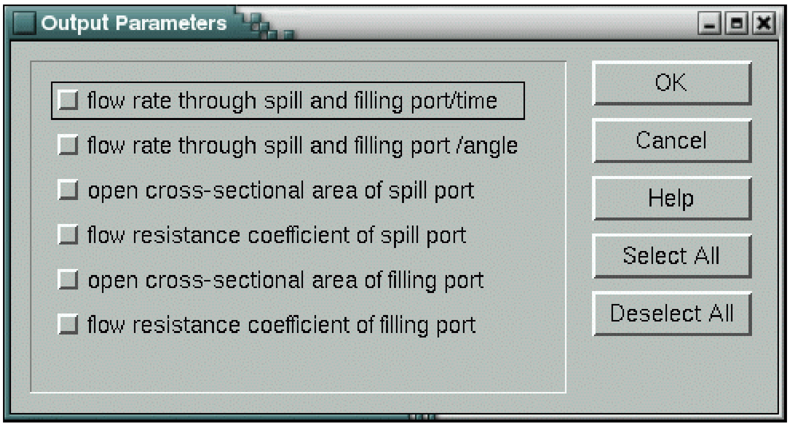


Figure 130: Output Parameters of Distributor Filling and Spill Ports Dialog

14.2.5. Flow Equations

Fluid flow through **Filling and Spill Ports of Distributor Pump** is calculated according to gate valve model in a straight tube[27] as shown in Figure 131.

[27] Idelchik, I.E., *Handbook of Hydraulic Resistance,* Springer, 2nd Edition, 1986.

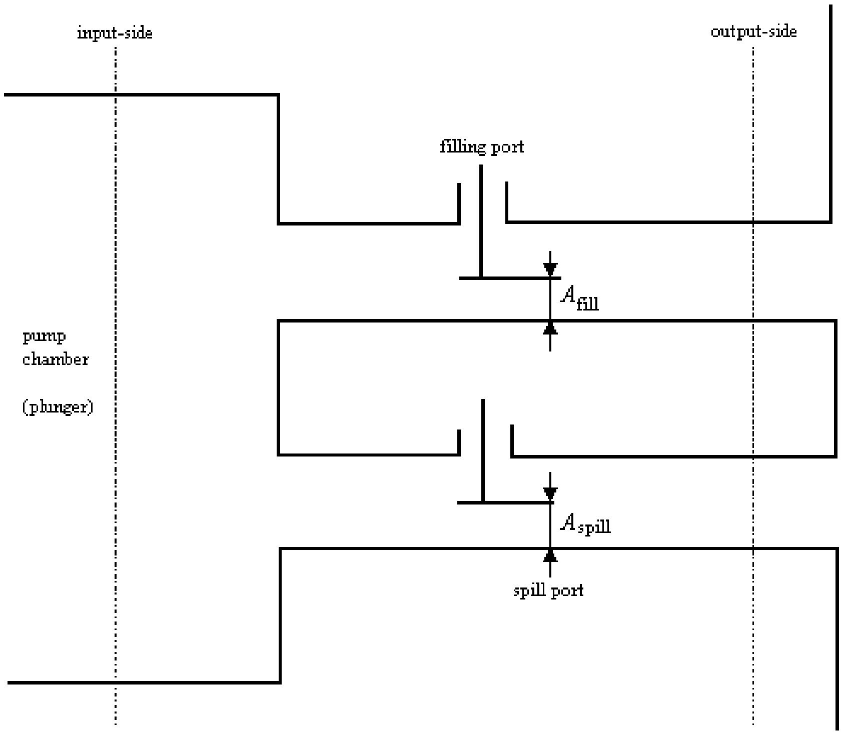


Figure 131: Flow Model through Filling and Spill Ports of Distributor Pump

Flow rate through filling and spill ports is calculated from Bernoulli equation:

$$\dot{Q}_{fill} = sign(\Delta p)\, A_{fill} \sqrt{\frac{1}{\left(\frac{A_{fill}}{A_{fill\_port}}\right)^2 - \left(\frac{A_{fill}}{A_{plunger}}\right)^2 + \xi_{fill}} \frac{2|\Delta p|}{\rho}}, \tag{257}$$

$$\dot{Q}_{i\_spill} = sign(\Delta p)\, A_{spill} \sqrt{\frac{1}{\left(\frac{A_{spill}}{A_{spill\_port}}\right)^2 - \left(\frac{A_{spill}}{A_{plunger}}\right)^2 + \xi_{spill}} \frac{2|\Delta p|}{\rho}}, \tag{258}$$

with:

$\dot{Q}_{fill}$	flow rate through filling port
$\dot{Q}_{i\_spill}$	flow rate through ith spill port
$\Delta p = p_{in} - p_{out}$	pressure difference between pump inlet and outlet
A_{fill}	minimum flow area (cross-section area of plunger groove or calculated flow area at fill port)
A_{spill}	calculated flow area at spill port
A_{fill_port}	area of fully open fill port
A_{spill_port}	area of fully open spill port

$A_{plunger}$	plunger cross-sectional area
ξ_{fill}	Program-calculated of user-defined flow resistance of fill port
ξ_{spill}	Program-calculated of user-defined flow resistance of spill port

Total flow rate is given by:

$$\dot{Q} = \dot{Q}_{fill} + N_{s\_port} \cdot \dot{Q}_{i\_spill}, \tag{259}$$

where N_{s_port} is number of spill ports (all must be identical).

Areas A_{fill} and A_{spill} are calculated from geometrical characteristic of the pump position of plunger in x-direction and actual cam angle. If there is no input connection for rotation, cam angle is calculated from actual time step and Reference speed defined in **Calculation Control**.

A_{spill} ... calculated flow area at spill port

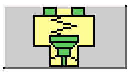
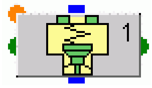
$$x_{plung} \le X_{eff} \qquad A_{spill} = 0$$

$$x_{plung} > X_{eff} \qquad A_{spill} = b_{spill}\left(x_{plung} - X_{eff}\right)$$

$$x_{plung} > X_{eff} + h_{spill} \qquad A_{spill} = b_{spill} h_{spill}$$

15. VALVE

15.1. Delivery Valve

Element Name:	**Delivery Valve**	
Element Icon:		
Definition:	This element serves to define a Delivery Valve. It can be used standalone or as a part of Snubber Valve or Constant Pressure Valve.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection must be specified.	

☞

Note: Special pins provide information about position and velocity of valve body in local x-direction for connected elements via special connection.

Note: To model a typical **Snubber Valve, Delivery Valve** has to be switched in parallel with **Orifice** element. To model a **Constant Pressure Valve**, **Delivery Valve** has to be switched in parallel with a **Pressure Check Valve** (spherical or conical).

Note: **Delivery Valve** allows fluid flow only in one (delivery) direction: from input end to output end. Hence the hydraulic connections to a **Delivery Valve** are irreversible: input connection implies pressure under valve seat (pump side) and output connection – pressure above valve seat (injector side).

The schematic of **Delivery Valve** is shown in Figure 132.

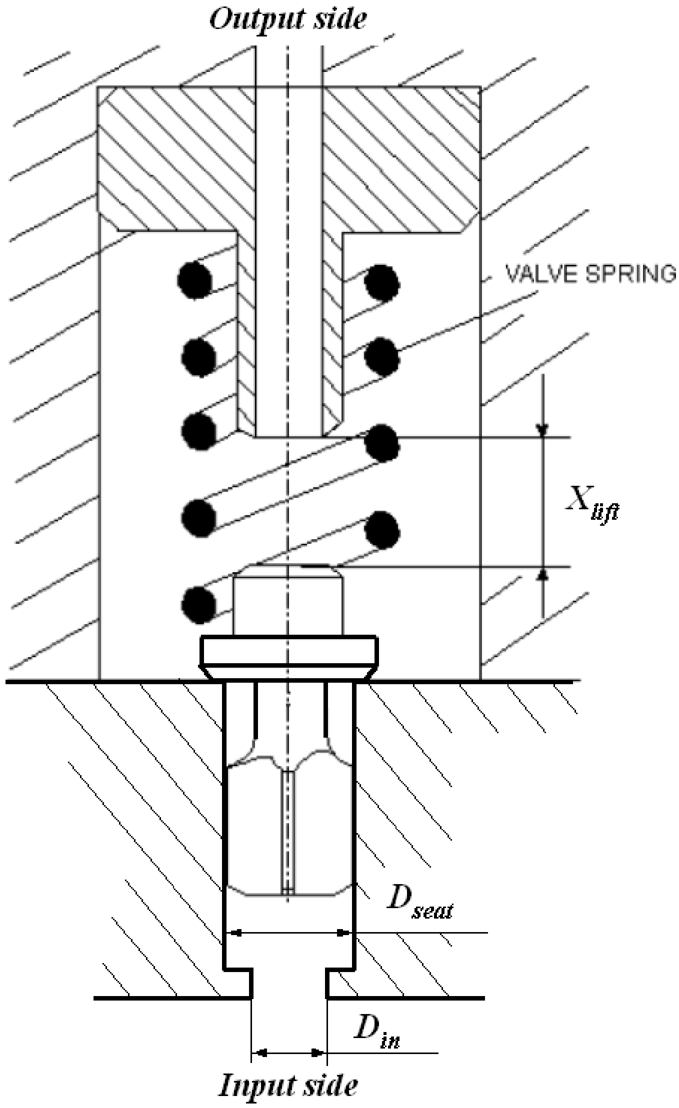


Figure 132: Delivery Valve - Geometry

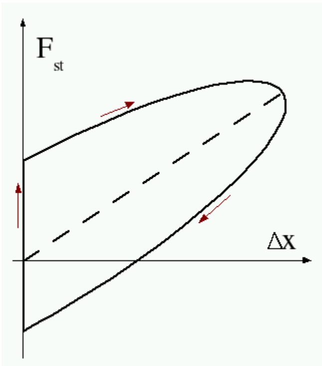
15.1.1. Input Parameters

Description of input data for the **Delivery Valve**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Moving mass	Specify moving mass *m*. It is equal to the valve body mass plus 33% of the total mass of the attached springs (connections).
Maximum lift of valve body	Specify maximum lift (stroke) of valve body X_{lift} (refer to Figure 132). **Note:** If button is not active, Valve stop data is faded out.
Valve seat diameter	Specify diameter of valve seat D_{seat} (refer to Figure 132).

Fluid damping at valve closing	Specify viscous damping coefficient at valve closing c_{fluid} (optional).
Flow resistance coefficient at valve seat	Specify flow resistance coefficient ξ related to narrowest cross-sectional flow area of valve seat at fully opened valve.
Opening /closing pressure (+/-)	Specify opening/closing pressure (p_0). To define the closing pressure valve (spring on input side) for opening/closing pressure negative value has to be entered.
VALVE THROTTLES	
Diameter of input throttle	Specify diameter of throttle on input side D_{in} (before valve seat).
Diameter of output throttle	Specify diameter of throttle on output side D_{out} (after seat). Note: output throttle is optional; if exists, it has to be before the Delivery valve volume (do not mix with outlet pipe at output side)
Flow resistance coefficient at input throttle	Specify flow resistance coefficient of input throttle ξ_{in} (before valve seat) related to cross-sectional area of the input throttle. If no better data are available, ξ_{in} can be estimated from Table 10 located in Chapter 19.1..
Flow resistance coefficient at output throttle	Specify flow resistance coefficient of output throttle ξ_{out} (after valve seat) related to cross-sectional area of the output throttle. ξ_{out} is required only if output throttle diameter D_{out} is defined (nonzero). If no better data are available, ξ_{out} can be estimated from Table 10 located in Chapter 19.1.
VALVE ELEMENTS	
Valve spring stiffness	Specify the stiffness constant of valve spring k_0.
Valve spring damping	Specify the damping constant of valve spring c_0.
USER-DEFINED FLOW AREA AT SEAT	
Geomteric Seat Flow Area	Specify coordinate (lift) of valve body in ascending order in the first column. Specify geometric cross-sectional flow area of valve seat in the second column. Effective cross-sectional flow area of **Delivery Valve** seat is calculated by multiplying geometric area with flow discharge coefficient at valve seat General rules for the specification of table data: • The intermediate values are bridged by linear interpolation. • If the actual body lift exceeds the maximum value in the table, the last specified value of flow area will be used • If the actual body lift is less than the lift specified in the first table row, the first specified value of flow area will be used.
VALVE BODY SEAT/STOP	
(Linear or Pseudo-linear Stop model)	

Valve body seat stiffness	Specify the valve body seat stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop stiffness	Specify the valve body stop stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat damping	Specify the valve body seat damping on input end (c_{in_st}). Damping is active when valve body lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Valve body stop damping	Specify the valve body stop damping on output end (c_{out_st}). Damping is active when valve body lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
(Nonlinear Stop model)	
Valve body seat linear stiffness	Specify the valve body seat linear stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop linear stiffness	Specify the valve body stop linear stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat Stiffness factor	Specify the valve body seat stiffness factor on input end (n_{in_st}).
Valve body stop Stiffness factor	Specify the valve body stop stiffness factor on output end (n_{out_st}).
Valve body seat Elastic impact coef.	Specify the valve body seat elastic impact coefficient on input end (α_{in_st}).
Valve body stop Elastic impact coef.	Specify the valve body stop elastic impact coefficient on output end (α_{out_st}).
Valve stop stiffness	Specify the stiffness constant of valve stop k_{out_st}.
Valve stop damping	Specify the damping constant of valve stop c_{out_st}.

Seat/Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**).  **Figure 133: Linear Stop Model**

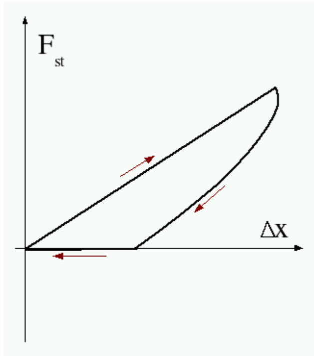
<table>
<tr><td>Seat/Stop model is pseudo-linear</td><td>This model is derived from the linear model by introducing additional switching function (see equations 65 and 68).
The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact.


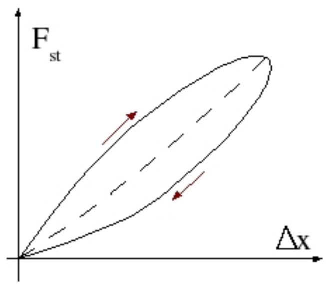
Figure 134: Pseudo-linear Stop Model</td></tr>
<tr><td>Stop model is non-linear</td><td>Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model is used.


Figure 135: Non-linear Stop Model</td></tr>
</table>

DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **0**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and force as result of valve spring deformation ($c_0\dot{x} + k_0x + F_0$).	

15.1.2. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial conditions:

x-coordinate	Specifies the valve body initial position of valve body in local x-direction.
velocity in x-direction	Specifies the valve body initial velocity of valve body in local x-direction.

All initial values that are not specified under initials are set to 0.

15.1.3. Modify Parameter

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

Coulomb friction force	SI Unit:	N
flow resistance coefficient at valve seat	SI Unit:	---
valve opening pressure	SI Unit:	Pa (N/m^2)
stiffness of valve spring	SI Unit:	N/m
damping of valve spring	SI Unit:	Ns/m

maximum lift of valve body	SI Unit:	m
fluid damping at valve closing	SI Unit:	Ns/m

15.1.4. Output Parameters

To activate output parameter, the check box in front of output parameter must be checked.

Description of output parameters:

x-coordinate of valve body	The valve body position is calculated from Equation 260 described in Section 15.1.5.
velocity of valve body in x-direction	The valve body velocity is calculated from Equation 260 described in Section 15.1.5.
volumetric flow rate through valve	Volumetric flow rate through valve is calculated from Equation 268. Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter **26.2.6.3**). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
narrowest open cross-sectional area at valve seat	Cross-sectional flow area at valve seat is calculated from the Equation 269 (refer to Section 15.1.5).
flow resistance coefficient at valve seat	Same as flow resistance coefficient at valve seat specified in **Input Data** dialog box.
Forces are calculated in local x-direction:	
resultant force	Total force acting on valve body (refer to Equation 260).
hydraulic force	Hydraulic force is calculated from Equation 262.
mechanical force	Mechanical force is the sum of all spring and seat/stop forces.

15.1.5. Equations of Motion

Motion of **Delivery Valve** body in local coordinate system is governed by the following equation:

$$m\ddot{x} + c_0\dot{x} + k_0 x = -F_0 - F_{frict} - F_{hyd} - F_{damp} - F_{in\_st} - F_{out\_st} \qquad (260)$$

where m is the valve body mass, x is the body local coordinate, c_0 and k_0 are the damping and stiffness constants of the valve spring, F_0 is the preload force of the valve spring, F_{frict} is Coulomb friction force, F_{hyd} is the hydraulic force, F_{in_st} and F_{out_st} are additional forces from input and output stops and F_{damp} is the damping force from squeezing fluid at valve closing. Forces are defined in local x-direction.

Preload force F_0 is calculated from the valve opening/closing pressure:

$$F_0 = p_0 \frac{\pi}{4} D_{seat}^2 , \tag{261}$$

where p_0 is the opening/closing pressure and D_{seat} is the valve seat diameter. Hydraulic force F_{hyd} is calculated from the equation:

$$F_{hyd} = \left(p_{out} - p_{in}\right)\frac{\pi}{4} D_{seat}^2 , \tag{262}$$

where p_{in} and p_{out} are the actual pressures on input and output end. Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries). Fluid damping force at valve closing F_{damp} is defined as follows:

$$\begin{aligned} &\forall\ \dot{x} < 0 : F_{damp} = c_{fluid}\dot{x} , \\ &\forall\ \dot{x} \ge 0 : F_{damp} = 0, \end{aligned} \tag{263}$$

where c_{fluid} is the viscous damping coefficient of squeezing fluid. Obviously, F_{damp} is always zero at valve opening (upward motion out of seat) and, if $c_{fluid} \neq 0$, nonzero at valve closing (downward motion to seat).

Force from input stop F_{in_st} is defined as follows:

Linear stop model

$$\begin{aligned} &\forall\ x < 0 : F_{in\_st} = c_{in\_st}\dot{x} + k_{in\_st}x , \\ &\forall\ x \ge 0 : F_{in\_st} = 0, \end{aligned} \tag{264}$$

Pseudo-linear stop model

$$\begin{array}{llcl} \forall x < 0 & & & \\ & \forall \dot{x} < 0 & : & F_{in\_st} = k_{in\_st}x , \\ & \forall \dot{x} \ge 0 & : & F_{in\_st} = c_{in\_st}\dot{x} + k_{in\_st}x , \\ & \forall F_{in\_st} > 0 & : & F_{in\_st} = 0 , \\ \forall x \ge 0 & & & \\ & & : & F_{in\_st} = 0 , \end{array} \tag{265}$$

where c_{in_st} and k_{in_st} are the damping and stiffness coefficients of the input stop.

Similarly, force from output stop F_{out_st} is given by:

Linear stop model

$$\begin{aligned} &\forall\ x \le X_{lift} : F_{out\_st} = 0 , \\ &\forall\ x > X_{lift} : F_{out\_st} = c_{out\_st}\dot{x} + k_{out\_st}\left(x - X_{lift}\right), \end{aligned} \tag{266}$$

Pseudo-linear stop model

$$
\begin{array}{lll}
\forall x \le X_{lift} & & \\
 & : & F_{out\_st} = 0\,, \\
\forall x > X_{lift} & & \\
\quad \forall \dot{x} > 0 & : & F_{out\_st} = k_{out\_st}\left(x - X_{lift}\right), \\
\quad \forall \dot{x} \le 0 & : & F_{out\_st} = c_{out\_st}\dot{x} + k_{out\_st}\left(x - X_{lift}\right), \\
\quad \forall \mathrm{F_{out\_st}} < 0 & : & F_{out\_st} = 0\,,
\end{array}
\qquad (267)
$$

where c_{out_st} and k_{out_st} are the damping and stiffness coefficients of the output stop and X_{lift} is the valve body stroke (maximum lift).

Volumetric flow rate through **Delivery Valve** is calculated from Bernoulli equation:

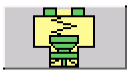
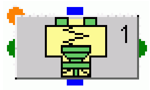
$$\dot{Q} = A_{seat}\sqrt{\frac{1}{\xi_{seat}}\frac{2\left(p_{in} - p_{out}\right)}{\rho}} + \frac{\pi}{4}D_{seat}^{2}\dot{x}\,, \qquad (268)$$

where $\dot{Q}$ is the flow rate, A_{seat} is the narrowest open cross-sectional area at valve seat, ξ_{seat} is the flow resistance coefficient at seat and ρ is the fluid density.

If the option “User-defined flow are at seat” is active, then cross-sectional flow area A_{seat} is interpolated from the user-defined table according to the actual valve body lift. If the option “User-defined flow are at seat” is switched off (default case), then flat valve seat geometry is assumed. For the flat seat, area A_{seat} is calculated by:

$$A_{seat} = \pi D_{seat} x\,. \qquad (269)$$

15.2. Constant Volume Valve

Element Name:	**Constant Volume Retraction Valve**	
Element Icon:		
Definition:	This element serves to define a Constant Volume Delivery Valve with a retraction piston.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection must be specified.	

☞

Note: Special pins provide information about position and velocity of valve body in local x-direction for connected elements via special connection.

Note: **Constant Volume Valve** allows fluid flow only in one (delivery) direction: from input to output end. Hence, the hydraulic connections to a Constant Volume Valve are irreversible. Input connection implies pressure under valve seat (pump side) and output connection – pressure above valve seat (injector side).

The schematic of **Constant Volume Valve** is shown in Figure 136.

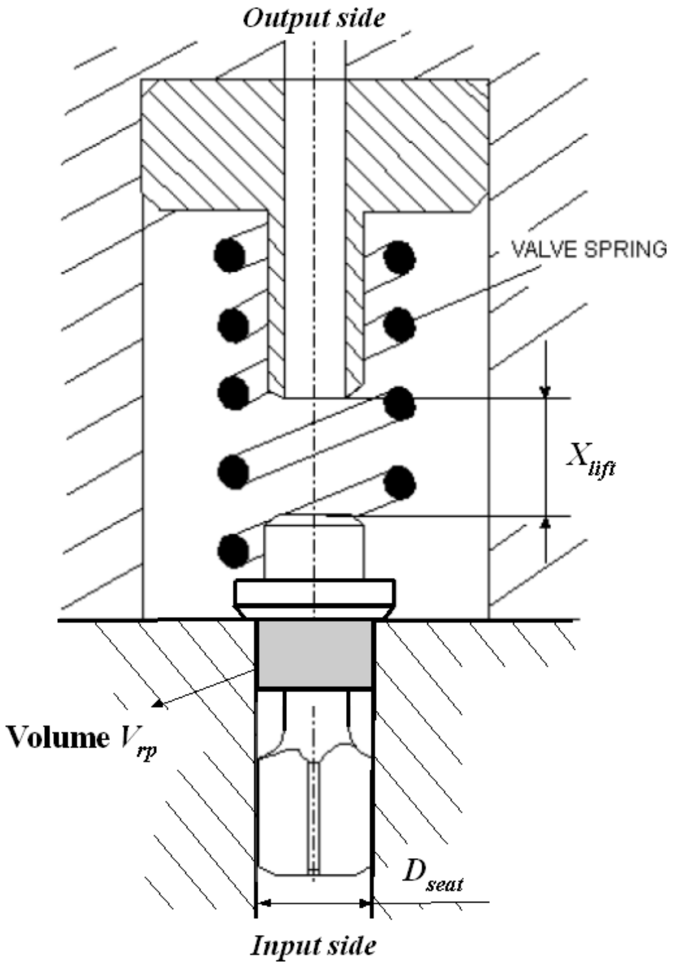


Figure 136: Constant Volume Valve - Geometry

15.2.1. Input Parameters

Description of input data for the **Constant Volume Valve**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Moving mass	Specify moving mass *m*. It is equal to the valve body mass plus 33% of the total mass of the attached springs (connections).

Maximum lift of valve body	Specify maximum lift (stroke) of valve body X_{lift}. (refer to Figure 136). **Note:** If button is not active, Valve stop data is faded out.
Valve seat diameter	Specify diameter of valve seat D_{seat} (refer to Figure 136) which is also the diameter of reraction piston.
Retraction (unloading) volume	Specify size of retraction volume V_{rp} (refer to Figure 136).
Coulomb friction force	Specify the Coulomb friction force in local x-direction F_{frict}.
Fluid damping at valve closing	Specify viscous damping coefficient at valve closing c_{fluid}.
Flow resistance coefficient at valve seat	Specify flow resistance coefficient ξ_{seat} related to narrowest cross-sectional flow area of valve seat at opened valve.
Opening pressure	Specify pressure difference for valve opening p_0.
VALVE ELEMENTS	
Valve spring stiffness	Specifies the stiffness constant of valve spring k_0.
Valve spring damping	Specifies damping constant of valve spring c_0.
VALVE BODY SEAT/STOP	
(Linear or Pseudo-linear Stop model)	
Valve body seat stiffness	Specify the valve body seat stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop stiffness	Specify the valve body stop stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat damping	Specify the valve body seat damping on input end (c_{in_st}). Damping is active when valve body lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Valve body stop damping	Specify the valve body stop damping on output end (c_{out_st}). Damping is active when valve body lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
(Nonlinear Stop model)	
Valve body seat linear stiffness	Specify the valve body seat linear stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop linear stiffness	Specify the valve body stop linear stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat Stiffness factor	Specify the valve body seat stiffness factor on input end (n_{in_st}).
Valve body stop Stiffness factor	Specify the valve body stop stiffness factor on output end (n_{out_st}).

Valve body seat Elastic impact coef.	Specify the valve body seat elastic impact coefficient on input end (α_{in_st}).
Valve body stop Elastic impact coef.	Specify the valve body seat elastic impact coefficient on output end (α_{out_st}).
Seat/Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**) (refer to **Figure 54**).
Seat/Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function (see equations **65** and **68**). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact (refer to **Figure 55**).
Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model is used.
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **0**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and force as result of valve spring deformation ($c_0\dot{x} + k_0 x + F_0$).	

15.2.2. Initial Conditions

Initial Conditions of **Constant Volume Valve** are identical to the **Initial Conditions** of **Delivery Valve** described in Section 15.1.2.

15.2.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

Coulomb friction force	SI Unit:	N
flow resistance coefficient at valve seat	SI Unit:	---
valve opening pressure	SI Unit:	Pa (N/m^2)
stiffness of valve spring	SI Unit:	N/m
damping of valve spring	SI Unit:	Ns/m
maximum lift of valve body	SI Unit:	M
fluid damping at valve closing	SI Unit:	Ns/m

15.2.4. Output Parameters

Path: **Element**| **Store Results**

To activate output parameter, the check box in front of output parameter must be checked.

Description of output parameters:

x-coordinate of valve body	The valve body position is calculated from Equation 270 as described in Section 15.2.5.
velocity of valve body in x-direction	The valve body velocity is calculated from Equation 270 as described in Section 15.2.5.
volumetric flow rate through valve	Volumetric flow rate through valve is calculated from Equation 271. Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter 26.2.6.3). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
narrowest open cross-sectional area	Cross-sectional flow area at valve seat is calculated from the Equation 272.
flow resistance coefficient at valve seat	Same as flow resistance coefficient at valve seat ξ specified in **Input Data** dialog box.
Forces are calculated in local x-direction:	
resultant force	Total force acting on valve body (refer to Equation 270).
hydraulic force	Hydraulic force is calculated from Equation 262

mechanical force	Mechanical force is the sum of all spring and seat/stop forces.

15.2.5. Equations of Motion

Motion of **Constant Volume Valve** body in local coordinate system is governed by the following equation:

$$m\ddot{x} + c_0\dot{x} + k_0 x = -F_0 - F_{frict} - F_{hyd} - F_{damp} - F_{in\_st} - F_{out\_st}, \quad (270)$$

where m is the valve body mass, x is the body coordinate, c_0 and k_0 are the damping and stiffness constants of the valve spring, F_0 is the preload force of the valve spring, F_{frict} is Coulomb friction force, F_{hyd} is the hydraulic force, F_{in_st} and F_{out_st} are additional forces from input and output stops and F_{damp} is the damping force of squeezing fluid at valve closing. Forces are defined in local x-direction.

Preload force F_0 and hydraulic force F_{hyd} are calculated in the same way as for the **Delivery Valve** (refer to Equations 261 and 262). Also forces from input and output stops F_{in_st} and F_{out_st} are calculated identically to those of the **Delivery Valve** (refer to equations 264 and 265). Fluid damping force F_{damp} is calculated from Equation 263.

Volumetric flow rate through **Constant Volume Valve** is calculated from the equation:

$$\dot{Q} = A_{seat}\sqrt{\frac{1}{\xi_{seat}}\frac{2(p_{in} - p_{out})}{\rho}} + \frac{\pi}{4}D_{seat}^2\dot{x}, \quad (271)$$

where:

$\dot{Q}$	flow rate
A_{seat}	narrowest open cross-sectional area at valve seat
ξ_{seat}	flow resistance coefficient at valve seat
p_{in} and p_{out}	actual pressures on input and output end
ρ	fluid density
D_{seat}	diameter of valve seat (retraction piston)
$\dot{x}$	velocity of valve body

Flow area A_{seat} depends on the vvalve body lift and is defined by:

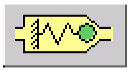
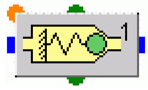
$$\begin{aligned} &\forall x \le h_{rp} : \; A_{seat} = 0, \\ &\forall x > h_{rp} : \; A_{seat} = \pi D_{seat}(x - h_{rp}), \end{aligned} \quad (272)$$

where h_{rp} is the length (height) of the retraction piston. It is calculated from the retraction volume V_{rp} by the formula:

$$h_{rp} = \frac{4V_{rp}}{\pi D_{seat}^2}. \tag{273}$$

Equation of motion of **Constant Volume Valve** body (Equation 270) is identical to the equation of motion of **Delivery Valve** body (Equation 260). However, the calculation of the flow rate is different (refer to Equations 271 and 268). Due to retraction piston there is an additional effect produced at the valve closing. It enlarges the volume of the injection line by the size of the retraction volume thus causing a sharp pressure drop in the line. Therefore retraction volume has to be adjusted to the length of the injection line.

15.3. Check Valve (Ball)

Element Name:	**Check Valve** with spherical body	
Element Icon:		
Definition:	This element serves to define a Pressure Check Valve with spherical body (ball).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection must be specified.	

Note: Special pins provide information about position and velocity of valve body in local x-direction for connected elements via special connection.

Note: **Check Valve** allows fluid flow in one direction only: from input to output end (refer to Figure 137). Hence the hydraulic connections to a **Check Valve** are irreversible, i.e. input and output connections cannot be interchanged.

Note: To model a **Constant Pressure Valve, Check Valve** has to be switched in parallel to a **Delivery Valve** in such a way that its hydraulic connection s are opposite to the hydraulic connections of the **Delivery Valve**. In this case, input connection of the **Check Valve** must be on the injector side and output connection on the pump side.

The schematic of **Check Valve** with spherical body is shown in Figure 137.

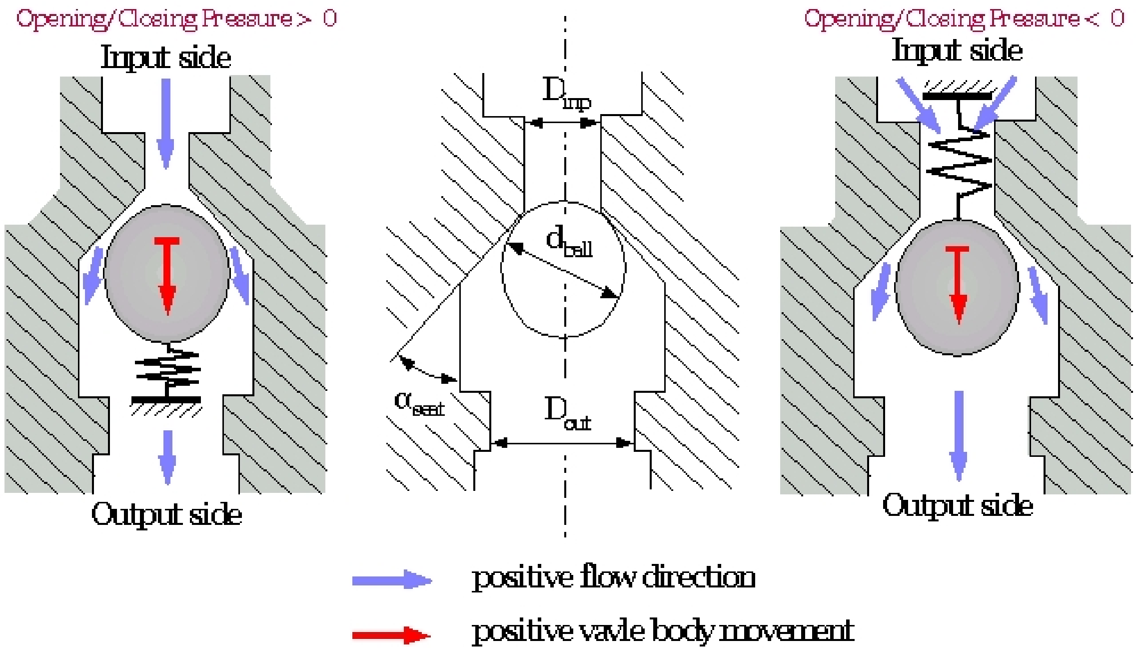


Figure 137: Model of Check Valve with Spherical Body

15.3.1. Input Parameters

Description of input data for the **Check Valve (Ball)**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Solid Properties …	Press this bar to define the solid properties of the valve body and seat. Global solid properties are active by default.

☞ **Note:** Necessary for calculation of Hertz stress between the ball and conical seat.

Moving mass	Specify moving mass *m*. It is equal to the valve body mass plus 33% of the total mass of the attached springs (connections).
Diameter of valve ball	Specify valve ball diameter d_{ball} (refer to Figure 137)
Maximum lift of valve ball	Specify maximum lift (stroke) of valve body X_{lift}. **Note:** If button is not active, Valve stop data is faded out.
Half-angle of valve seat	Specify half-angle of valve seat α_{seat} (refer to Figure 137)
Fluid damping at valve closing	Specify viscous damping coefficient at valve closing c_{fluid}.

Flow resistance coefficient at valve seat	Specify flow resistance coefficient ξ_{seat} at narrowest open cross-sectional area of valve seat. If no data is available for ξ_{seat}, it can be estimated from Figure 139.
Opening /closing pressure (+/-)	Specify pressure difference for valve opening (p_0). To define the closing pressure valve (spring on input side) for opening/closing pressure negative value has to be entered.
VALVE THROTTLES	
Diameter of input throttle	Specify diameter of throttle on input end (valve seat) D_{in} (refer to Figure 137).
Diameter of output throttle	Specify diameter of throttle on output end D_{out}. (refer to Figure 137).

☞ **Note:** D_{in} and D_{out} may not have to be defined in Input Data dialog box (i.e. they are left 0). In this case, the respective flow resistances will not be calculated (no throttles).

Flow resistance coefficient at input throttle	Specify flow resistance coefficient through throttle on input end ξ_{in} (to valve seat) related to cross-sectional area of the input throttle. ξ_{in} is required only if the input throttle diameter D_{in} is defined (nonzero). If no better values are available, ξ_{in} can be estimated from Table 10 located in Chapter 19.1..
Flow resistance coefficient at output throttle	Specify flow resistance coefficient through throttle on output end ξ_{out} (from valve seat) related to cross-sectional area of the output throttle ξ_{out} is required only if output throttle diameter D_{out} is defined (nonzero). If no better values are available, ξout can be estimated from Table 10 located in Chapter 19.1.
VALVE ELEMENTS	
Valve spring stiffness	Specifies stiffness constant of valve spring k_0.
Valve spring damping	Specifies the damping constant of valve spring c_0.
VALVE BODY SEAT/STOP	
(Linear or Pseudo-linear Stop model)	
Valve body seat stiffness	Specify the valve body seat stiffness on input end (k_{in_st}). Stiffness is active when valve body lift x < 0.
Valve body stop stiffness	Specify the valve body stop stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat damping	Specify the valve body seat damping on input end (c_{in_st}). Damping is active when valve body lift x < 0 (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Valve body stop damping	Specify the valve body stop damping on output end (c_{out_st}). Damping is active when valve body lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).

(Nonlinear Stop model)	
Valve body seat linear stiffness	Specify the valve body seat linear stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop linear stiffness	Specify the valve body stop linear stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat Stiffness factor	Specify the valve body seat stiffness factor on input end (n_{in_st}).
Valve body stop Stiffness factor	Specify the valve body stop stiffness factor on output end (n_{out_st}).
Valve body seat Elastic impact coef.	Specify the valve body seat elastic impact coefficient on input end (α_{in_st}).
Valve body stop Elastic impact coef.	Specify the valve body stop elastic impact coefficient on output end (α_{out_st}).
Seat/Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**) (refer to **Figure 54**).
Seat/Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function (see equations **65** and **68**). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact (refer to **Figure 55**).
Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model is used.
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **0**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and force as result of valve spring deformation ($c_0\dot{x} + k_0 x + F_0$).	

15.3.2. Initial Conditions

Initial Conditions of **Check Valve** with spherical body are identical to the **Initial Conditions** of **Delivery Valve** (refer to Section 15.1.2).

15.3.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

fluid damping at valve closing	SI Unit:	Ns/m
diameter of input side throttle	SI Unit:	m
flow resistance coefficient of input throttle	SI Unit:	---
diameter of output side throttle	SI Unit:	m
flow resistance of output throttle	SI Unit:	---
flow resistance coefficient at valve seat	SI Unit:	---
valve opening pressure	SI Unit:	Pa (N/m^2)
stiffness of valve spring	SI Unit:	N/m
damping of valve spring	SI Unit:	Ns/m
maximum lift of valve ball	SI Unit:	m

15.3.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

x-coordinate of valve ball	The valve body position is calculated according to Equation 274.
velocity of valve ball in x-direction	The valve body velocity is calculated according to Equation 274.
volumetric flow rate through valve	Volumetric flow rate through valve is calculated from Equation 277. Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter **26.2.6.3**). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
narrowest open cross-sectional area at valve seat	Valve seat cross-sectional flow area is calculated from the Equation 279.
discharge coefficient related to narrowest flow area at seat	Discharge coefficient related to seat flow area is calculated value according to Equation 280.
resultant force	Total force acting on valve body (refer to Equation 274).
hydraulic force	Hydraulic force is calculated from Equation 276.
mechanical force	Mechanical force is the sum of all spring and seat/stop forces.

15.3.5. Equations of Motion

Motion of **Check Valve** ball in local coordinate system is governed by the following equation:

$$m\ddot{x} + c_0\dot{x} + k_0 x = -F_0 - F_{hyd} - F_{frict} - F_{in\_st} - F_{out\_st}, \qquad (274)$$

where m is the ball mass, x is the ball local coordinate, c_0 and k_0 are the damping and stiffness constants of the valve spring, F_0 is the preload force of the valve spring, F_{hyd} is the hydraulic force, F_{in_st} and F_{out_st} are additional forces from input and output stops and F_{damp} is the damping force of squeezing fluid at valve closing. Forces are defined in local x-direction.

Preload force F_0 is calculated from the valve opening/closing pressure:

$$F_0 = p_0 \frac{\pi}{4}\left(d_{ball} \cos\alpha_{seat}\right)^2, \qquad (275)$$

where p_0 is the opening/closing pressure, d_{ball} is the ball diameter and α_{seat} is the half-angle of valve seat.

Hydraulic force F_{hyd} is calculated from the equation:

$$F_{hyd} = \frac{\pi}{4}\left(p_{out} - p_{in}\right)\left(d_{ball} \cos\alpha_{seat}\right)^2, \qquad (276)$$

where p_{in} and p_{out} are the actual pressures on input and output end. Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries). Fluid damping force is calculated from Equation 263.

Forces from input and output stops F_{in_st} and F_{out_st} are calculated identically to those of the **Delivery Valve** (refer to Equations 264 and 265).

Volumetric flow rate through **Check Valve** is calculated from Bernoulli equation:

$$\dot{Q} = \sqrt{\frac{1}{\xi_{eq}} \frac{2\left(p_{in} - p_{out}\right)}{\rho}}, \qquad (277)$$

where $\dot{Q}$ is the flow rate, ξ_{eq} is the equivalent flow resistance coefficient and ρ is the fluid density. ξ_{eq} is obtained from the relationship:

$$\xi_{eq} = \frac{\xi_{in}}{A_{in}^2} + \frac{\xi_{seat}}{A_{seat}^2} + \frac{\xi_{out}}{A_{out}^2}, \qquad (278)$$

where ξ_{in} and ξ_{out} are the flow resistance coefficients of input and output throttles, A_{in} and A_{out} are the cross-sectional areas of the respective throttles and A_{seat} is the narrowest open cross-sectional area at valve seat. Flow area A_{seat} is given by:

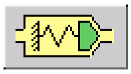
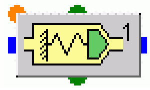
$$\begin{aligned} &\forall\, x \le 0 : A_{seat} = 0, \\ &\forall\, x > 0 : A_{seat} = \pi x \sin\alpha_{seat} \cos\alpha_{seat}\left(d_{ball} + x \sin\alpha_{seat}\right). \end{aligned} \qquad (279)$$

Discharge coefficient μ_{seat} related to the narrowest cross-sectional flow area at valve seat is calculated from the formula:

$$\mu_{seat} = \frac{\sqrt{\frac{1}{\xi_{eq}}}}{A_{seat}}. \qquad (280)$$

It can be observed from the formulas (278) and (280) that if $\xi_{in} = \xi_{out} = 0$ (no throttles) then $\mu_{seat} = \sqrt{\frac{1}{\xi_{seat}}}$.

15.4. Check Valve (Poppet)

Element Name:	**Check Valve** with conical body	
Element Icon:		
Definition:	This element serves to define a constant Pressure Check Valve with conical body.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection must be specified.	

☞

Note: Special pins provide information about position and velocity of valve body in local x-direction for connected elements via special connection.

Note: **Check Valve** allows fluid flow in one direction only: from input to output end (refer to Figure 138). Hence the hydraulic connections to a **Check Valve** are irreversible, i.e. cannot be interchanged.

Note: To model a **Constant Pressure Valve**, **Check Valve** has to be switched in parallel to a **Delivery Valve** in such a way that its hydraulic connection s are opposite to the hydraulic connections of the **Delivery Valve**. In this case, input connection of the **Check Valve** must be on the injector side and output connection on the pump side.

The schematic of a **Check Valve** with conical body is shown in Figure 138.

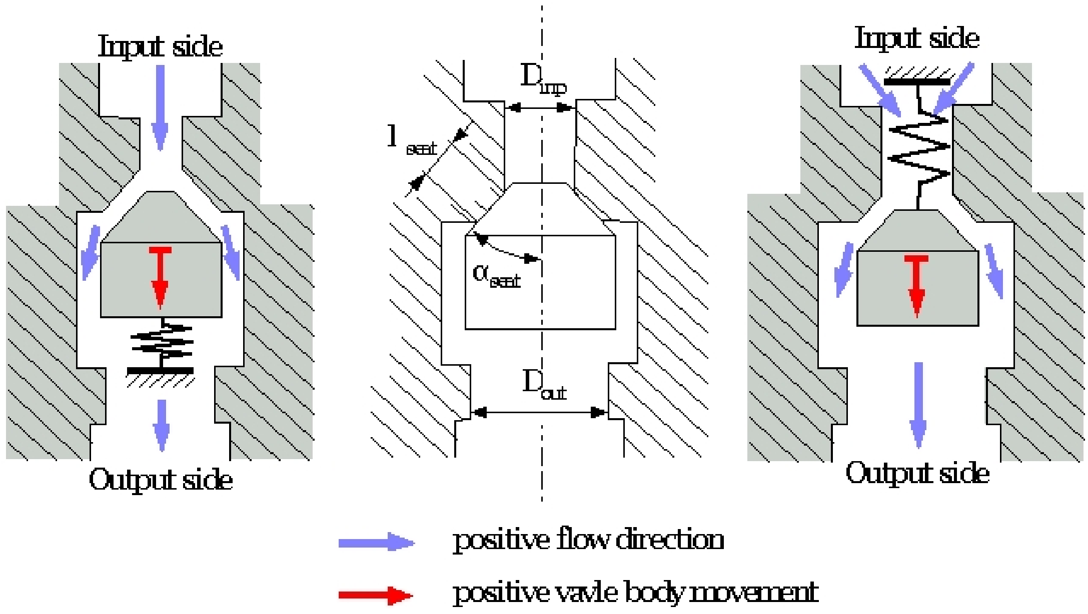


Figure 138: Model of Check Valve with Conical Body

15.4.1. Input Parameters

Description of input data for the **Check Valve (Poppet)**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Moving mass	Specify moving mass *m*. It is equal to the valve body mass plus 33% of the total mass of the attached springs (connections). However, mass *m* may be set to 0 (no inertia) if no mechanical vibration has to be considered.
Maximum lift of valve body	Specify maximum lift (stroke) of valve body X_{lift}. **Note:** If button is not active, Valve stop data is faded out.
Flow resistance coefficient at valve seat	Specify flow resistance coefficient ξ_{seat} at narrowest open cross-sectional area of valve seat.

Flow resistance coefficient ξ is related to discharge coefficient μ by the following formula:

$$\xi = \frac{1}{\mu^2}$$

If no data is available for the discharge coefficient μ at valve seat, it can be estimated from the diagrams in Figure 139 and Figure 140. The flow discharge coefficient μ (as a function of $\sqrt{\mathrm{Re}}$ where Re is Reynold's number) for zero seat length and different seat angles is depicted in Figure 139, and for seat angle 90° and different seat lengths in Figure 140[28].

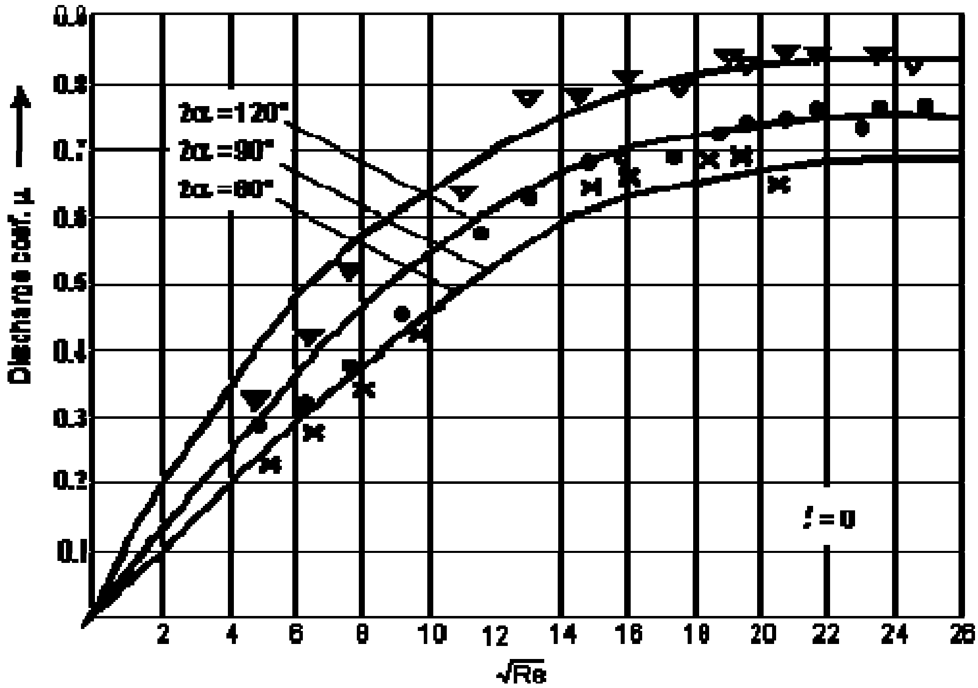


Figure 139: Function $\mu = f\left(\sqrt{\mathrm{Re}}\right)$ **(** $l = 0$ **). Discharge coefficient at valve seat for different seat angles (seat length** $l = 0$ **)**

[28] Kollek, W.; Kudzma, Z.; *Untersuchung des Einflusses von Konstruktionsparameter auf Stroemungersheinunugen in Sitzventilen mit kegelfoermigen Sperrsystem* KONSTRUKTION 40 1988, pp. 267-271.

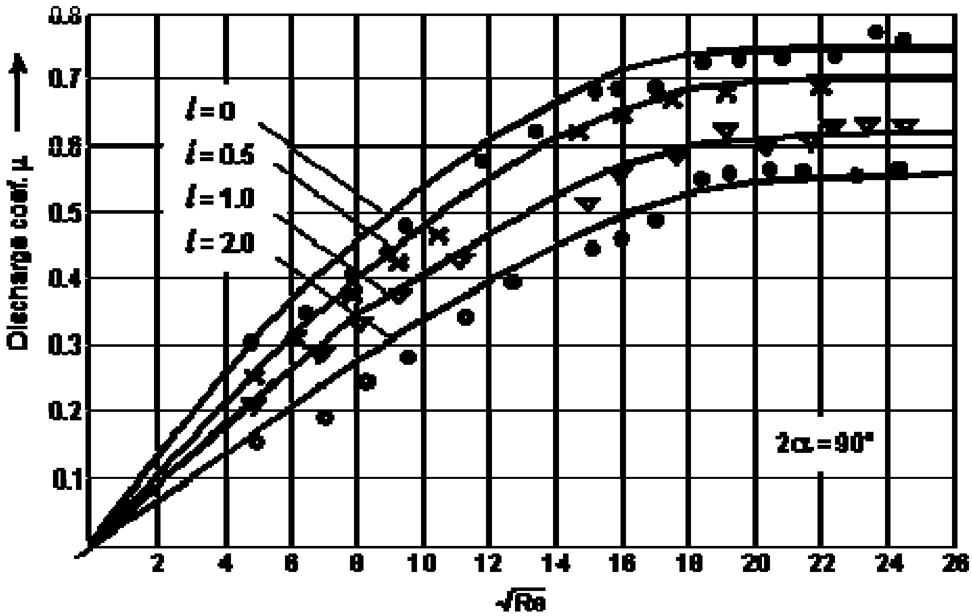


Figure 140: Function $\mu = f(\sqrt{Re}$ Discharge coefficient at valve seat for different seat length (seat angle 2α=90°)

Fluid damping at valve closing (optional)	Specify viscous damping coefficient at valve closing c_{fluid}.
Opening /closing pressure (+/-)	Specify pressure difference for valve opening (p_0). To define the closing pressure valve (spring on input side) for opening/closing pressure negative value has to be entered.
VALVE SEAT	
Seat diameter	Specify diameter of the valve seat D_{seat}. Refer to Figure 138.
Half-angle	Specify half-angle of valve seat α_{seat} (refer to Figure 138).
VALVE THROTTLES	
Diameter at input end	Specify diameter of throttle on input end D_{inp}.
Diameter at output end	Specify diameter of throttle on output end D_{out}.

☞ **Note:** D_{inp} and/or D_{out} may not be defined in **Input Data Dialog Box** (left 0). In this case,the flow resistance on input and/or output end will not be calculated.

Flow resistance coefficient at input end	Specify flow resistance coefficient in throttle on input end ξ_{inp} (to valve seat) related to cross-sectional area of the input throttle. ξ is required only if input throttle diameter D_{in} is defined (nonzero).

Flow resistance coefficient on output end	Specify flow resistance coefficient through throttles on output end ξ_{out} (from the valve seat) related to cross-sectional area of the output throttle. ξ_{out} is required only if output throttle diameter D_{out} is defined (nonzero). If better values are not available, ξ_{in} and ξ_{out} can be estimated from Table 10 located in Chapter 19.1..
VALVE ELEMENTS	
Valve spring stiffness	Specifies stiffness constant of the valve spring k_0.
Valve spring damping	Specifies the damping constant of valve spring c_0.
VALVE BODY SEAT/STOP	
(Linear or Pseudo-linear Stop model)	
Valve body seat stiffness	Specify the valve body seat stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop stiffness	Specify the valve body stop stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat damping	Specify the valve body seat damping on input end (c_{in_st}). Damping is active when valve body lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Valve body stop damping	Specify the valve body stop damping on output end (c_{out_st}). Damping is active when valve body lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
(Nonlinear Stop model)	
Valve body seat linear stiffness	Specify the valve body seat linear stiffness on input end (k_{in_st}). Stiffness is active when valve body lift $x < 0$.
Valve body stop linear stiffness	Specify the valve body stop linear stiffness on output end (k_{out_st}). Stiffness is active when valve body lift $x > X_{lift}$.
Valve body seat Stiffness factor	Specify the valve body seat stiffness factor on input end (n_{in_st}).
Valve body stop Stiffness factor	Specify the valve body stop stiffness factor on output end (n_{out_st}).
Valve body seat Elastic impact coef.	Specify the valve body seat elastic impact coefficient on input end (α_{in_st}).
Valve body stop Elastic impact coef.	Specify the valve body stop elastic impact coefficient on output end (α_{out_st}).
Seat/Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**) (refer to **Figure 54**).
Seat/Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function (see equations **65** and **68**). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact (refer to **Figure 55**).

Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model is used.
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **0**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and force as result of valve spring deformation ($c_0\dot{x} + k_0 x + F_0$).	

15.4.2. Initial Conditions

Initial Conditions of **Check Valve** with conical body are identical to the **Initial Conditions** of **Delivery Valve**.

15.4.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section 26.2.3.2). Open by pressing **Modify...** bar.

Modifiable parameters:

fluid damping at valve closing	SI Unit:	Ns/m
diameter of throttle bore on input end	SI Unit:	m
flow resistance coefficient of input throttle	SI Unit:	---
diameter of throttle bore on output end	SI Unit:	m
flow resistance coefficient of output throttle	SI Unit:	---

flow resistance coefficient at valve seat	SI Unit:	---
valve opening pressure	SI Unit:	Pa (N/m^2)
stiffness of valve spring	SI Unit:	N/m
damping of valve spring	SI Unit:	Ns/m
maximum lift of valve body	SI Unit:	m

15.4.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

x-coordinate of valve body:	The valve body position is calculated according to Equation 281 as described in Section 15.4.5.
velocity of valve body in x-direction:	The valve body velocity is calculated according to Equation 281 as described in Section 15.4.5.
volumetric flow rate through valve:	Volumetric flow rate through valve is calculated from Equation 277. Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve:	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** input dialog box (refer to Chapter 26.2.6.3). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
narrowest open cross-sectional area at valve seat:	Valve seat cross-sectional flow area is calculated from the Equation 284 (refer to Section 15.4.5).
discharge coefficient related to narrowest flow area at seat	Discharge coefficient is calculated from Equation 280.
Forces are calculated in local x-direction:	
resultant force	Total force acting on valve body (refer to Equation 281).
hydraulic force	Hydraulic force is calculated from Equation 283.
mechanical force	Mechanical force is the sum of all spring and seat/stop forces.

☞ **Note:** Output parameter list of the **Check Valve** with conical body is the same as the output parameter list of the **Check Valve** with spherical body.

15.4.5. Equations of Motion

Motion of **Check Valve** body in local coordinate system is governed by the following equation:

$$m\ddot{x} + c_0\dot{x} + k_0 x = -F_0 - F_{hyd} - F_{damp} - F_{in\_st} - F_{out\_st}\ , \tag{281}$$

where m is the body mass, x is the body local coordinate, c_0 and k_0 are the damping and stiffness constants of the valve spring, F_0 is the preload force of the valve spring, F_{hyd} is the hydraulic force, F_{in_st} and F_{out_st} are additional forces from input and output stops and F_{damp} is the damping force of squeezing fluid at valve closing. Forces are defined in local x-direction.

Preload force F_0 is calculated from the valve opening/closing pressure:

$$F_0 = p_{open}\frac{\pi}{4}D_{seat}^2\ , \tag{282}$$

where p_O is the opening/closing pressure and D_{seat} is the diameter of valve seat.

Hydraulic force F_{hyd} is calculated from the equation:

$$\begin{aligned} &\forall\ x < 0 : F_{hyd} = \frac{\pi}{4}\left(p_{out} - p_{in}\right)D_{seat}^2, \\ &\forall\ x \geq 0 : F_{hyd} = \pi\left(p_{out} - p_{in}\right)\left(\frac{D_{seat}}{2} - x\sin\alpha_{seat}\cos\alpha_{seat}\right)^2, \end{aligned} \tag{283}$$

where p_{in} and p_{out} are the actual pressures on input and output end and α_{seat} is the half-angle of valve seat. Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries). Fluid damping force F_{damp} is calculated from Equation 263.

Forces from input and output stops F_{in_st} and F_{out_st} are calculated identically to those of the **Delivery Valve** (refer to Equations 264 and 265).

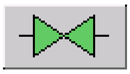
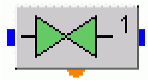
Volumetric flow rate through **Check Valve** with conical body is calculated by the same Bernoulli Equation 277 as the flow rate through **Check Valve** with a ball. However, the narrowest open cross-sectional flow area at valve seat A_{seat} is obtained from the following equation:

$$\begin{aligned} &\forall\ x \leq 0 : A_{seat} = 0, \\ &\forall\ x > 0 : A_{seat} = \pi x\sin\alpha_{seat}\left(D_{seat} - x\sin\alpha_{seat}\cos\alpha_{seat}\right). \end{aligned} \tag{284}$$

Equivalent flow resistance coefficient ξ_{eq} and discharge coefficient μ_{seat} related to the narrowest cross-sectional flow area at valve seat are calculated in the same way as for the **Check Valve** with spherical body (refer to Equations 278 and 280).

16. THROTTLE

16.1. Time-Controlled Throttle (Switch Valve)

Element Name:	**Switch Valve**
Element Icon:	
Definition:	This element serves to define a throttle controlled by timing of flow areas (switch valve). Flow area can be a function of time or crank angle.
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection can be specified. 

☞ **Note:** Switch Valve can be typically used for modeling of a two-way solenoid or other type of control valve (as the simplest model without consideration of the valve body dynamics).

16.1.1. Input Parameters

Description of input data for the **Switch Valve**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar has to specify the local fluid properties (refer to Section 8.1.4).
VALVE TIMING (START OF SWITCHING)	
Initial state:	
Closed	Depending on the chosen initial state different column order for Time/Crank Angle table will appear.
Open	Depending on the chosen initial state different column order for Time/Crank Angle table will appear.
Timing data:	
Absolute(T)	Switching events are defined in absolute time scale in the Time/Crank Angle (CA) table.
Relative(delta T)	Switching events are defined in relative time scale in the Time/Crank Angle (CA) table, i.e. each new switching event refers to the previous event.

Reference:	
Calculation Start	Switching events are referenced to Calculation start.
BTDC (firing)	Switching events are referenced to firing BTDC (time/angle Before Top Dead Center of firing). Active for Absolute Timing Data only.
Time/Crank Angle	Specify time/angle at which the valve opening and closing events start.

EFFECTIVE CROSS-SECTIONAL FLOW AREA my*A	
Scaling factor for 2nd column (my)	This button activates the multiplication factor for 2nd column of valve opening table. Values in second column will be multiplied with scale factor as soon as **Run /Run Sets/Restart** is clicked in **Simulation** pull-down menu. It is convenient to use **Scaling factor for 2nd column** as discharge coefficient μ (if it is constant and valve closing/opening areas are available in absolute values not pre-multiplied by μ).

☞ **Note:** Multiplication by factor is performed not online but only at **Run /Run Sets/Restart** command. Hence the displayed μA table at valve opening will remain unchanged irrespective of the value of scaling factor.

my*A of valve seat at opening	Specify time values of valve opening in ascending order in first column. Specify values of effective cross-sectional flow area of valve seat at opening in ascending order in second column. General rules for the specification: Intermediate values are bridged by linear interpolation. Values in the first table row must be 0./0. Opening function has to be monotonously increasing. Last effective cross-sectional flow area (μA) in opening table has to be equal to first one in closing table.
Scaling factor for 2nd column (my)	This button activates the multiplication factor for 2 nd column of valve closing table. Values in second column will be multiplied with scale factor as soon as **Run /Run Sets/Restart** is clicked in **Simulation** pull-down menu. It is convenient to use **Scaling factor for 2nd column** as discharge coefficient (if it is constant and seat closing/opening areas are available in absolute values not pre-multiplied by μ).

☞ **Note:** Multiplication by factor is performed not online but only at **Run /Run Sets/Restart** command. Hence the displayed μA table at valve opening will remain unchanged irrespective of the value of scaling factor.

my*A of valve seat at closing	Specify time values of valve closing in ascending order in first column. Specify values of effective cross-sectional flow area of valve seat at closing in descending order in second column. General rules for the specification of table data: • Intermediate values are bridged by linear interpolation. • Time value in the first table row and area μA value in the last row must be 0. • Closing function has to be monotonously decreasing. • First effective cross-sectional flow area (μA) in closing table has to be equal to the last area in opening table. • If the first switching event T1 is closing, then the μA value in the first row has to be equal to effective cross sectional flow area of valve seat at start of calculation (time=0). Refer to the description of opening characteristic for details on entering data.

☞ **Note:** **Switch Valve** can have only one opening and one closing characteristic (only one area corresponding to each time/angle). Therefore **valve opening curve must** be **monotonously increasing** and **valve closing curve** must be **monotonously decreasing**. For modeling of more complex opening/closing characteristics, intermediate switching with multiple **Switch Valve** elements or **Variable Throttle** element has to be used.
Effective cross-sectional flow area is kept constant in time interval when the previous switching procedure is finished and the next one has not yet started.

16.1.2. Initial Conditions

Initial conditions cannot be specified for **Time-Controlled Throttle** (**Switch Valve**).

16.1.3. Modify Parameter

Modifiable parameters cannot be specified for **Time-Controlled Throttle** (**Switch Valve**).

16.1.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, check the option box in front of output parameter.

Description of output parameters:

volumetric flow rate through valve	Volumetric flow through **Switch Valve** is calculated from the Equation 285 as given in Section 16.1.5. Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain.
cumulative flow rate through valve	Cumulative flow rate per calculation range.
effective cross-sect. valve flow area	Flow area μA is calculated from valve opening or closing characteristic according to the actual time.

16.1.5. Flow Equation

Volumetric flow rate through **Switch Valve** is calculated from Bernoulli equation:

$$\dot{Q} = sign(p_{in} - p_{out})\mu A\sqrt{\frac{2|p_{in} - p_{out}|}{\rho}} \tag{285}$$

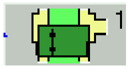
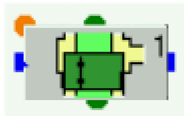
where:

$\dot{Q}$	flow rate
μA	effective cross-sectional flow area (at valve seat)
p_{in} and p_{out}	actual pressures on input and output end
ρ	fluid density

Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries).

Note that Equation 285 contains a *sign* function is necessary to determine the flow direction depending on the pressures on input and output.

16.2. Lift-Controlled Throttle (Slide Valve)

Element Name:	**Slide Valve**
Element Icon:	
Definition:	This element serves to define a throttle controlled by a position of one or two mechanical bodies. Flow area is a function of absolute or relative lift.
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection can be specified. 

☞

Note: **Slide Valve** alone is not a physical element but basically a control function. Hence, it needs a link to a mechanical body (piston-type element). Mechanical body has to be connected to **Slide Valve** by special connection (green line). With this connection, **Slide Valve** will receive information about the position of the body.

Note: If **Slide Valve** is connected via two special connections to two mechanical bodies (piston-type elements), its flow area is a function of the coordinate difference (relative lift) between these two elements. In this case the mechanical bodies must be interconnected via mechanical connection.

16.2.1. Input Parameters

Description of input data for the **Slide Valve**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar has to specify the local fluid properties (refer to Section 8.1.4).
EFFECTIVE CROSS-SECTIONAL FLOW AREA (my*A)	
Factor for 2nd column (my)	This button activates the multiplication factor for 2 nd column (flow area). For alternative option refer to **Constant Flow discharge coef. (** μ **)** at the dialog bottom.. Values in second column will be multiplied with scaling factor as soon as **Run/Run Sets/Restart** is selected in the **Simulation pull-down menu**. It is convenient to use **Scaling factor for 2nd column** as discharge coefficient μ (if it is constant and valve closing and opening areas are available in absolute values not pre-multiplied by μ) .

Flow Area	Specify coordinate (lift) of mechanical body (or relative lift between two bodies) in ascending order in the first column. Specify effective cross-sectional flow areas of valve seat in the second column. Cross-sectional flow area of **Slide Valve** is calculated according to the actual position of the mechanical body. General rules for the specification of table data: • The intermediate values are bridged by linear interpolation. • If the actual body lift exceeds the maximum value in the table, the last specified value of flow area is used for the further calculation. • If the actual body lift is less than the lift specified in the first table row, the first specified value of flow area will be used. Refer to the description of tables in **Switch Valve** in Section 16.1.1. for entering data.
Flow Discharge (my)	Specify flow discharge coefficient (μ) for respective area row.

Constant flow discharge coef. (my)	
Discharge coef. for forward flow	Specify the flow discharge coefficient for the forward flow μ_{in} (from input to output end).
Discharge coef. for backward flow	Specify the flow discharge coefficient for the backward flow μ_{out} (from output to input end).

☞ **Note:** Option **Constant flow discharge coef. (** μ **)** allows to specify different flow discharge coefficients for the forward and backward flow. If it is activated, multiplication **Factor for 2nd column** of the **Effective Flow Area** table is automatically deactivated and 3rd table column (**Flow Discharge**) disabled.

16.2.2. Initial Conditions

Initial conditions cannot be specified for **Lift-Controlled Throttle** (**Slide Valve**).

16.2.3. Modify Parameter

Modifiable parameters cannot be specified for **Lift-Controlled Throttle** (**Slide Valve**).

16.2.4. Output Parameters

Path: **Element│ Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

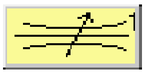
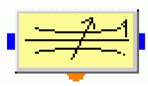
Description of output parameters:

volumetric flow rate through valve	Volumetric flow rate through **Slide Valve** is calculated from Equation 285 (refer to Section 16.1.5). Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain.
cumulative flow rate through valve	Cumulative flow rate per calculation range.
effective cross-sectional valve flow area	BOOST Hydsim calculates effective cross-sectional flow area from input table according to the actual lift of mechanical body linked to **Slide Valve** by special connection.
lift of connected body (absolute/relative)	If **Slide Valve** is connected via only one special connection to mechanical body (piston-type element), BOOST Hydsim takes coordinate of this element. If **Slide Valve** is connected via two special connections to two mechanical bodies (piston-type elements), BOOST Hydsim calculates coordinate difference (relative lift) between these two elements.

16.2.5. Flow Equation

Volumetric flow rate through **Slide Valve** is calculated by the same Bernoulli equation (Equation 285) as the flow rate through **Switch Valve** (refer to Section 16.1.5).

16.3. Flow Area vs. Time/CA

Element Name:	**Flow Area vs. Time/CA (Variable Throttle)**	
Element Icon:		
Definition:	This element serves to define a variable throttle with flow area as a function of time or crank angle.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection can be specified.	

16.3.1. Input Parameters

Description of input data for the **Flow Area vs. Time/CA (Variable Throttle)**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).

EFFECTIVE FLOW AREA (my*A)	
Shift Time/CA relative to calculation start	Specify the Shift time/crank angle for the first table column. It will be added to the time/crank angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Factor for 2nd column (my)	This button activates the multiplication factor for 2nd column (flow area). For alternative option refer to **Constant flow discharge coef. (** μ **)**at the dialog bottom. Values in second column will be multiplied with scaling factor as soon as **Run/Run Sets/Restart** is selected in the **Simulation pull-down menu**. It is convenient to use **Scaling factor for 2nd column** as discharge coefficient μ (if it is constant and valve closing and opening areas are available in absolute values not pre-multiplied by μ).
Copy data for each cycle (720 deg Crank angle):	Activate this button to specify the values in the table as a function of time (or crank angle) for one cycle only (720 degrees of Crank Angle). If more than two revolutions of the crankshaft are considered in the calculation, the program will copy data from the table for each new cycle.

☞ **Note:** If the Copy data button is **active** in the Simulation/Boundary Data global dialog for each cycle, then it will have dominant status and it will be active at Simulation/Run, Run Sets or Restart.

Flow Area	Specify time/crank angle in ascending order in first column. Specify effective cross-sectional flow areas of valve seat in second column. Effective cross-sectional flow area of **Variable Throttle** is calculated according to the actual time/crank angle. General rules for the specification of table data: • The intermediate values are bridged by linear interpolation. • If the actual time/CA exceeds the maximum value in the table, the last specified value of flow area is used for the further calculation. • If the actual time/CA is less than the lift specified in the first table row, the first specified value of flow area will be used. Refer to the description of tables in **Variable Throttle** in Section 16.1.1. for entering data.
Flow Discharge (my)	Specify flow discharge coefficient (μ) for respective area row. If active, **Factor for 2nd column** is disabled.

Constant flow discharge coef. (my)	
Discharge coef. for forward flow	Specify the flow discharge coefficient for the forward flow μ_{in} (from input to output end).
Discharge coef. for backward flow	Specify the flow discharge coefficient for the backward flow μ_{out} (from output to input end).

☞ **Note:** Option **Constant flow discharge coef. (** μ **)** allows to specify different flow discharge coefficients for the forward and backward flow. If it is activated, multiplication **Factor for 2nd column** of the **Effective Flow Area** table is automatically deactivated and 3rd table column (**Flow Discharge**) disabled.

16.3.2. Initial Conditions

Initial conditions cannot be specified for **Variable Throttle**.

16.3.3. Modify Parameter

Modifiable parameters cannot be specified for **Variable Throttle**.

16.3.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

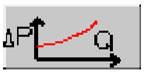
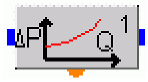
Description of output parameters:

volumetric flow rate through valve	Volumetric flow through **Variable Throttle** is calculated from the Equation 285 as given in Section 16.1.5. Volumetric flow rate through valve in **Time** domain.
volumetric flow rate through valve	Volumetric flow rate through valve in **Reference angle** domain or **Crank angle** domain.
cumulative flow rate through valve	Cumulative flow rate per calculation range.
effective cross-sectional valve flow area	BOOST Hydsim calculates effective cross-sectional flow area from the input table according to the actual **Time** or **Crank angle**.

16.3.5. Flow Equation

Flow rate through **Variable Throttle** is calculated by the same Bernoulli equation (Equation 285) as the flow rate through **Switch Valve** (refer to Section 16.1.5). If the option **Flow discharge coef. (** μ **)** is active, then $\mu = \mu_{in}$ for the forward flow (from input to output end) and $\mu = \mu_{out}$ for the backward flow (from output to input end).

16.4. Pressure Drop vs. Flow

Element Name:	**Pressure vs. Flow Rate (Pressure - Flow Function)**
Element Icon:	
Definition:	This element serves to define pressure-flow characteristic of a generalized „throttle“ (pressure difference as a function of flow rate).
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection can be specified. 

Note: Pressure – Flow Rate Throttle can be typically used to represent the characteristic of a hydraulic pump, leaf valve etc.

16.4.1. Input Parameters

Description of input data for the **Slide Valve**:

Element name	The name of the element is specified as default.
PRESSURE – FLOW CHARACTERISTIC	
Factor for 1st column	This button activates the multiplication factor for 1st column. Values in first column will be multiplied with scaling factor as soon as **Run/Run Sets/Restart** is selected in the **Simulation pull-down menu**.
Pressure – Flow Characteristic Table	Specify flow rate data in ascending order in the 1st column. Specify pressure difference data in the 2nd column. Pressure drop – flow rate has to be a monotonous function. General rules for the specification of table data: • The intermediate values are bridged by linear interpolation. • If actual pressure drop exceeds the maximum pressure drop value in the table, the maximum specified value of pressure drop is used for the further calculation. • If the actual pressure drop is less than minimum pressure drop value in the table, the minimum specified value of pressure drop is used for the further calculation.
Disable reverse flow (from output to input)	This option forbids the reverse flow (from output to input). If it is activated, then the flow rate will be set to zero (disabled) for negative pressure difference (backward flow).

16.4.2. Initial Conditions

Initial conditions cannot be specified for **Pressure-Flow Throttle** element.

16.4.3. Modify Parameter

Modifiable parameters cannot be specified for **Pressure-Flow Throttle** element.

16.4.4. Output Parameters

Path: **Element** | **Store Results**

The **Output Parameters Dialog Box** of **Pressure-Flow Throttle** is shown in Figure 141.

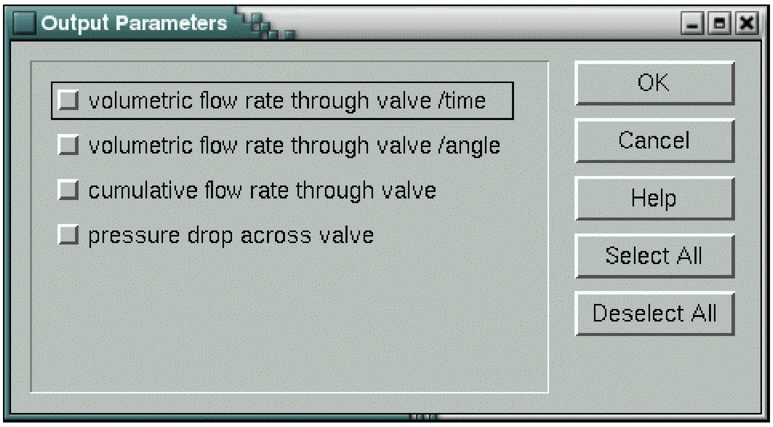
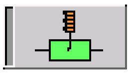
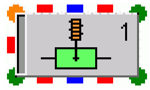


Figure 141: Output Parameters of Pressure-Flow Throttle Dialog

To activate output parameter, the check box in front of output parameter has to be checked.

17. SOLENOID

17.1. Armature (Basic model)

Element Name:	**Armature (Basic model)**	
Element Icon:		
Definition:	This element serves to define a solenoid armature actuated by a variable or „constant“ magnetic force. Variable magnetic force is a function of the armature lift.	
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 3 wire pins: 1	

Note: **Armature (Basic model)** element has only one degree-of-freedom (translation in x direction).

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **Armature (Basic model)** may have only one hydraulic connection on each end (input and output)

Note: Mechanical connections can be defined only in local x-direction of connection.

Note: For **Armature (Basic model)** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: Local x-axis of **Armature (Basic model)** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter: Direction of Element motion.

Note: For creating a model of the solenoid valve seat, an **Armature (Basic model)** element has to be used in combination with a **Slide Valve**.

Note: “Constant” magnetic force implies that actuation force is formally invariant of armature lift. In this simplified case the magnetic force is usually not constant but a time function. To define it, **Modify Parameter** option has to be used as described in Section 26.2.3.2.

Note: Hydraulic connections to **Armature (Basic model)** are irreversible, i.e. input connection always implies solenoid valve seat (armature lift at this position is 0) and output connection - output stop (maximum armature lift).

Schematic of a 2/2-way solenoid valve with armature and hydraulic connections is shown in Figure 142.

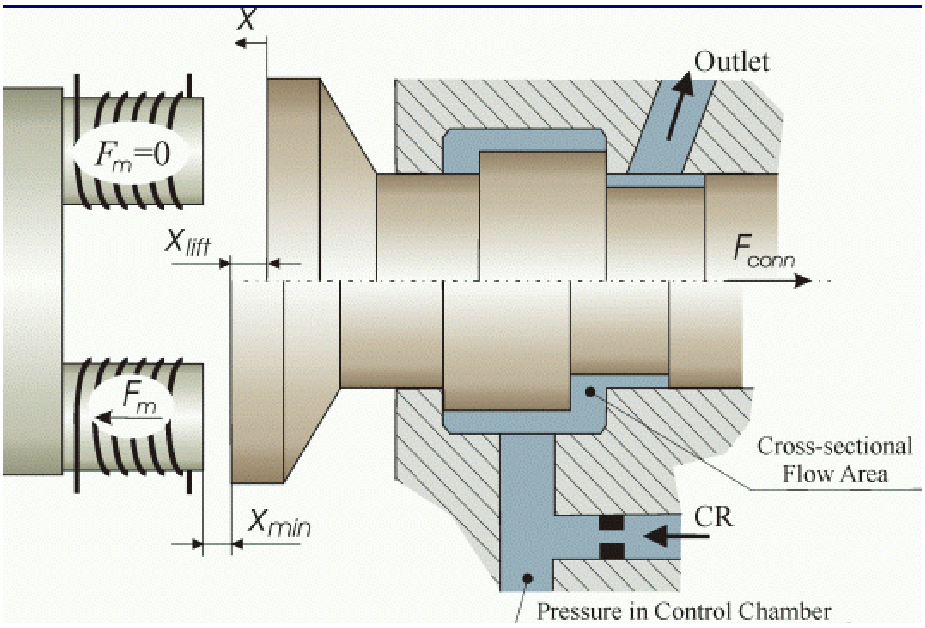


Figure 142: Schematic of a 2/2-Way Solenoid Valve with Two Armature Positions

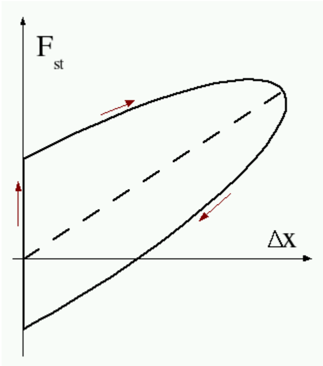
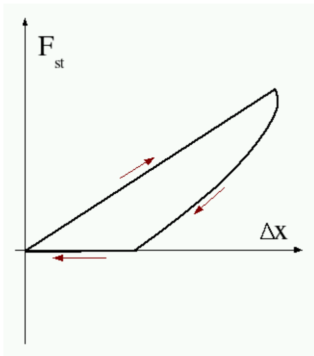
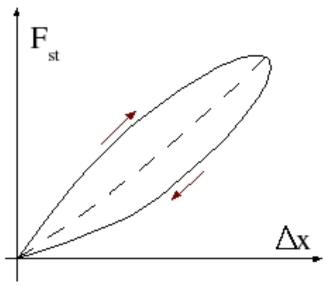
17.1.1. Input Parameters

Description of input data for the **Armature (Basic model)**:

Element name	The name of the element is specified as default.
Moving mass	Specify moving mass *m*. It has to be equal to the armature mass plus 33% of the mass of the connected springs.
Coulomb friction force	Specify constant dry friction force (if any).
Armature stroke (max. lift)	Specify distance between input and output stop (maximum lift X_{lift}). The coordinate of motion (*x*) has its origin at the side of the input stop.
MAGNETIC FORCE	
Constant/Function of time	
Force magnitude (maximum)	Specify the magnitude of constant magnetic force acting on armature when solenoid is switched on.
Function of gap and time	Specify the magnitude of magnetic force as a function of air gap between armature and pole (acting on armature when solenoid is switched on).
Min. air gap (to magnet pole)	Specify minimum air gap (x_{min}) between armature guide and solenoid pole. Note that x_{min} must be a positive value.

Max. Force Table	Specify air gap values in ascending order in the first column. Air gap x_{gap} is calculated from the formula: $x_{gap} = x_{min} + X_{lift} - x$, where x_{min} is the minimum air gap corresponding to the armature the position x=X_{lift}. Specify magnitude of magnetic force in the second column. Intermediate values are calculated by linear interpolation domain. **Note:** Air gap values outside the table range are undefined. If such a case occurs during calculation, an error message will be produced.
Force Correction Factor	Press this bar to specify magnetic force correction factor for one switching event. It can be used to account for magnetic force change due to the variation of actuation amperage and nonlinear magnetic field effects (e.g. hysteresis).
Switch On	
Force Correction Factor Table	Specify the magnetic force correction factor as a function of Time/Crank Angle. Time t=0 (first table row) implies a Switch On instant.
Switch Off	
Force Correction Factor Table	Specify the magnetic force correction factor as a function of Time/Crank Angle. Time t=0 (first table row) implies a Switch Off instant.
Further Armature (Valve Body) Data	Press this bar to specify the armature/valve body areas at input/output end and body stop stiffness and damping at input/output end.
Cross-section (under pressure)	
Body area	Specify the cross-sectional areas subjected to pressure.
Cross-sectional area at input end	Specify the cross-sectional area of armature/valve body at input end (seat).
Cross-sectional area at output end	Specify the cross-sectional area of armature/valve body at output end.
Body diameter	Specify the cross-section diameter subjected to pressure.
Cross-section diameter at input end	Specify the cross-section diameter of armature/valve body at input end (seat).
Cross-section diameter at output end	Specify the cross-section diameter of armature/valve body at output end.
If the input/output end of the **Armature** body is attached to **Volume** or **Hydraulic Boundary** element, the **Armature** input/output area will be under pressure. This will produce hydraulic forces which will enter the dynamic equation of motion of **Armature** as external forces. Furthermore, **Armature** velocity multiplied by input/output area will produce flow rate which will enter the continuity equation of the **Volume** element connected to the input/output end (refer to Sections **8.1.6** and **8.2.6**).	

Body stop data	
(Linear or Pseudo-linear Stop model)	
Body stop stiffness at input end	Specify the stiffness of armature/valve body on input end (k_{in_st}). Stiffness is active when piston lift x < 0.
Body stop stiffness at output end	Specify the stiffness of armature/valve body on output end (k_{out_st}). Stiffness is active when piston lift x > X_{lift}.
Body stop damping at input end	Specify the damping of armature/valve body on input end (c_{in_st}). Damping is active when piston lift x < 0 (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Body stop damping at output end	Specify the damping on of armature/valve body output end (c_{out_st}). Damping is active when piston lift x > X_{lif} (and for pseudo-linear stop model when $\dot{x} \leq 0$).
(Nonlinear Stop model)	
Body stop linear stiffness at input end	Specify the linear stiffness of armature/valve body on input end (k_{in_st}). Stiffness is active when piston lift x < 0.
Body stop linear stiffness at output end	Specify the linear stiffness of armature/valve body on output end (k_{out_st}). Stiffness is active when piston lift x > X_{lift}.
Body stop Stiffness factor at input end	Specify the stiffness factor of armature/valve body on input end (n_{in_st}).
Body stop Stiffness factor at output end	Specify the stiffness factor of armature/valve body on output end (n_{out_st}).
Body stop Elastic impact coef. at input end	Specify the elastic impact coefficient of armature/valve body on input end (α_{in_st}).
Body stop Elastic impact coef. at output end	Specify the elastic impact coefficient of armature/valve body on output end (α_{out_st}).

STOP MODEL	
Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive.  **Figure 143: Linear Stop Model**
Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function. The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact.  **Figure 144: Pseudo-linear Stop Model**
Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model is used.  **Figure 145: Non-linear Stop Model**

SOLENOID TIMING	
<u>Initial state:</u>	Depending on the initial state different column order in Time/Crank Angle table will appear.
Switched off	
Switched on	
<u>Timing data:</u>	Magnetic force switching events may be defined in absolute or relative Time/Crank Angle domain.
Absolute (T)	Magnetic force switching events are defined in absolute time scale in the Time/Crank Angle (CA) table.
Relative (delta T)	Magnetic force switching events are defined in relative time scale in the Time/Crank Angle (CA) table, i.e. each new switching event refers to the previous event.
<u>Reference:</u>	
Calculation Start	Magnetic force switching events are referenced to Calculation start.
BTDC (firing)	Magnetic force switching events are referenced to firing BTDC (time/angle Before Top Dead Center of firing). Active for Absolute Timing Data only.
<u>Time/Crank Angle (CA) Table</u>	Specify Time/Crank Angle instance at which magnetic force is switched on and off.
Data File	Specify data file from which the data will be loaded. This box becomes active when the Automatic Reload button is active.
Automatic Reload	If the Automatic Reload button is active, the file is automatically reloaded when the calculation is started. Automatic Reload button will not be faded out after table is loaded from file.
Use Default Data File Path	If the button Use Default Data File Path is active, the path name always starts in the directory where the AWS was started (all directory information is deleted from the Data File box, only the file name remains).
<u>Time Delay till force action</u>	
Delay from Switch ON	Specify the time delay from Switch On instant till magnetic force initiation.
Delay from Switch OFF	Specify the time delay from Switch Off instant till magnetic force initiation.
DIRECTION OF ELEMENT MOTION	
x(e1) **y(e2)** **z(e3)**	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and magnetic force (F_{magn}).	

17.1.2. Initial Conditions

Path: **Element**| **Initial Conditions**

All unspecified initial values are set to 0.

Description of initial conditions:

x-coordinate	Specify the initial position of the armature in local x-direction.
Velocity in x direction	Specify the initial velocity of the armature in local x-direction.

17.1.3. Modify Parameter

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

cross-sectional area at input end	SI Unit:	m^2
cross-sectional area at output end	SI Unit:	m^2
Coulomb friction force	SI Unit:	N
armature stroke (maximum lift)	SI Unit:	m
magnitude of magnetic force	SI Unit:	N
cross-section diameter at input end	SI Unit:	m
cross-section diameter at output end	SI Unit:	m

17.1.4. Output Parameters

Path: **Element**| **Store Results**

The **Output Parameters Dialog Box** of **Armature (Basic model)** is shown in Figure 146.

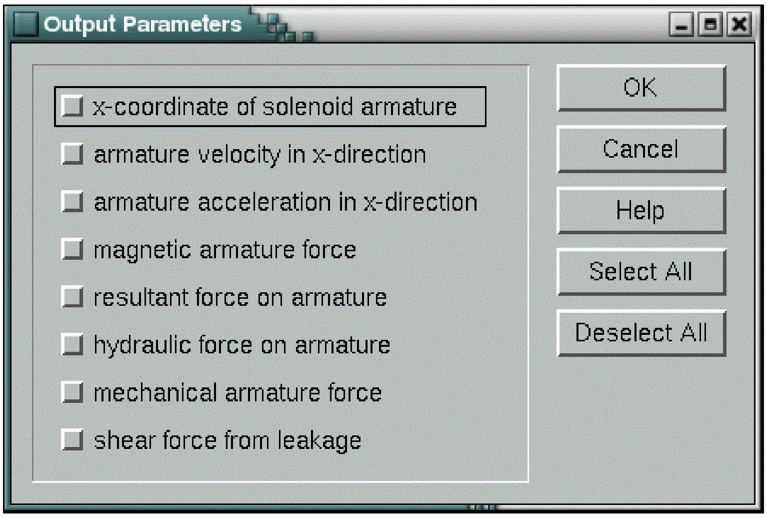


Figure 146: Output Parameters of Armature (Basic model) Dialog

To activate output parameter, the check box in front of output parameter has to be clicked.

Description of output parameters:

armature acceleration in x-direction	Armature coordinate and velocity are calculated according to Equation 286 as described in Section 17.1.5.
magnetic force	Magnetic force values are directly obtained from input (**Properties** or **Modify Parameter** dialogs) according to the solenoid timing (switching).

17.1.5. Equation of Motion

Motion of **Armature** in local coordinate system is governed by the following equation:

$$
\begin{aligned}
& m\ddot{x} \\
& +\sum_{i=1}^{n} c_i\left[\dot{x}\cos\left(\alpha-\alpha_{mc\_i}\right)-\dot{x}_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-\dot{y}_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
& +\sum_{i=1}^{n} k_i\left[x\cos\left(\alpha-\alpha_{mc\_i}\right)-x_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-y_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
& -\sum_{j=1}^{m} c_j\left[\dot{x}\cos\left(\alpha_{mcj}-\alpha\right)-\dot{x}_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+\dot{y}_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
& -\sum_{j=1}^{m} k_j\left[x\cos\left(\alpha_{mcj}-\alpha\right)-x_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+y_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
& -\sum_{i=1}^{n} F_{0i}\cos\left(\alpha-\alpha_{mc\_i}\right)+\sum_{j=1}^{m} F_{0j}\cos\left(\alpha_{mc\_j}-\alpha\right) \\
& = F_{magn}(x)-F_{hyd}-F_{frict}-F_{in\_st}-F_{out\_st}
\end{aligned}
\tag{286}
$$

where:

m	armature mass
x	local x-coordinate of armature
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends
Forces on right side of equation are defined in local x-direction:	
F_{magn}	magnetic force (lift-dependent or „constant“)
F_{hyd}	hydraulic force
F_{in_st} and F_{out_st}	additional forces from input and output stops
F_{frict}	Coulomb friction force

Hydraulic force F_{hyd} is calculated from the formula:

$$F_{hyd} = p_{out} A_{out} - p_{in} A_{in} \,, \tag{287}$$

where p_{in} and p_{out} are the input and output pressures and A_{in} and A_{out} are cross-sectional areas at input and output ends, respectively. Pressures p_{in} and p_{out} are taken from the connected hydraulic elements (volumes or hydraulic boundaries).

Gap-dependent magnetic force F_{magn} is defined as follows:

- for Switch ON event:

$$F_{magn}(x) = k_{ON}(t - t_{ON}) F_{\max}(x_{gap}) \,, \tag{288}$$

- for Switch OFF event:

$$F_{magn}(t) = k_{OFF}(t - t_{OFF})k_{ON}(t_{OFF})F_{\max}(x_{gap}), \tag{289}$$

where $k_{ON/OFF}$ is the time-dependent force correction factor and x_{gap} is the air gap between the armature guide and solenoid pole. Magnetic force F_{magn} during the Switch OFF event is product of last correction factor during the Switch ON event, actual correction factor for Switch OFF event and maximum magnetic force. Correction factor *k(t)* is separately defined for “Switch On” and “Switch Off” events.

The maximum value of magnetic force at each air gap value is directly picked from the magnetic force table. Intermediate values are calculated by linear interpolation. Note that no extrapolation is performed, i.e. any air gap value outside of the table range is invalid.

Air gap x_{gap} is calculated from the formula:

$$x_{gap} = x_{min} + X_{lift} - x\ , \tag{290}$$

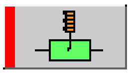
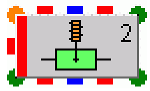
where X_{lift} is the armature stroke (distance between input and output stops), x is the armature lift and x_{min} is the minimum air gap between the armature guide and solenoid pole (gap at $x = X_{lift}$). It should be emphasized that the armature lift has its origin (zero position x=0) at the input stop (valve seat).

Equation **243** is necessary only for the calculation of the gap-dependent magnetic force.

Forces from input and output stops F_{in_st} and F_{out_st} are calculated in the same way as for the **Standard Piston**.

Equation 286 is highly nonlinear because magnetic force is inversely proportional to air gap or even air gap squared. Due to this it is difficult to solve and requires special care in specifying the integration step and solenoid parameters.

17.2. Armature (Extended Model)

Element Name:	**Armature (Extended model)**
Element Icon:	
Definition:	This element serves to define a solenoid armature actuated by a magnetic force. Magnetic force is function of the actuation amperage and armature lift.
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 3 wire pins: 1 

Note: **Armature (Extended model)** element has only one degree-of-freedom (translation in x direction).

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **Armature (Extended model)** may have only one hydraulic connection on each end (input and output)

Note: Mechanical connections can be defined only in local x-direction of connection.

Note: For **Armature (Extended model)** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: Local x-axis of **Armature (Extended model)** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter: Direction of Element motion.

Note: For creating a model of the solenoid valve seat, an **Armature (Extended model)** element has to be used in combination with a **Slide Valve**.

Note: Hydraulic connections to **Armature (Extended model)** are irreversible. Input connection always implies solenoid valve seat: at this position armature lift is 0 and air gap between the armature guide and magnet pole is maximum. Output connection implies an output stop where the armature lift is maximal and air gap is minimal.

Note: Magnetic force in **Armature (Extended model)** can be calculated internally by BOOST Hydsim or defined externally in a 3D-table as a function of the actuation amperage and armature lift.

Schematic of a 3/2-way solenoid valve with armature and hydraulic connections is shown in Figure 147.

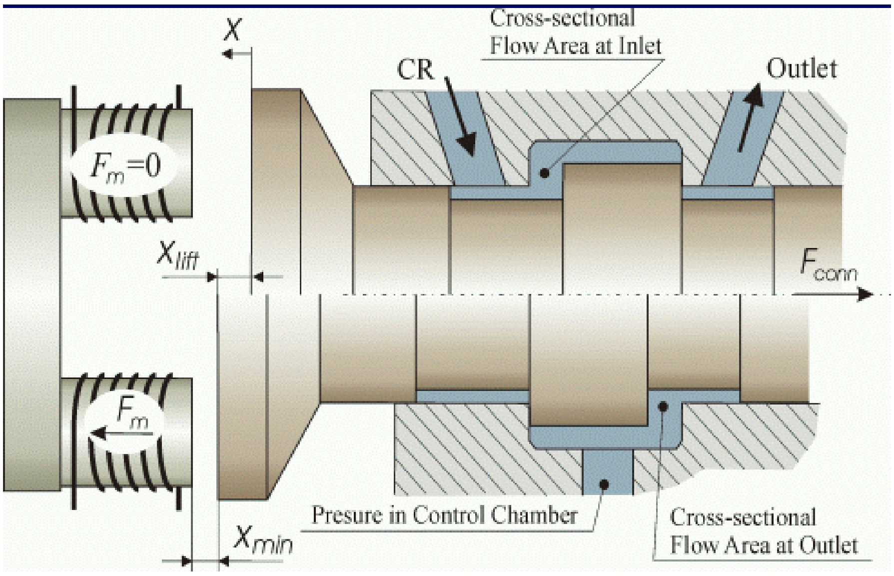


Figure 147: Schematic of a 3/2-Way Solenoid Valve with Two Armature Positions

17.2.1. Input Parameters

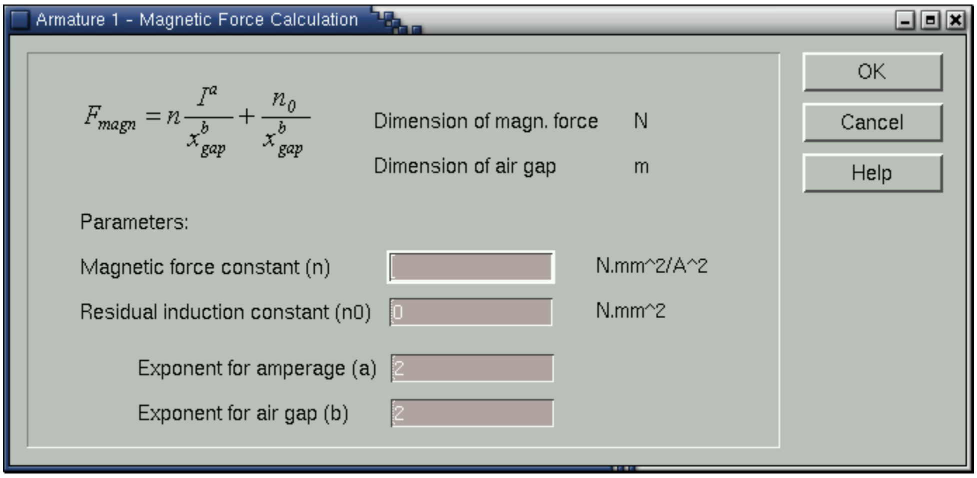
Path: **Element** | **Initial Values**

Description of input data for the **Armature (Extended model)**:

Element name	The name of the element is specified as default.
Moving mass	Specify moving mass *m*. It has to be equal to the armature mass plus 33% of the mass of the connected springs.
Coulomb friction force	Specify constant dry friction force (if any).
Armature stroke (max. lift)	Specify distance between input and output stops (maximum lift). The coordinate of motion (*x*) has its origin at the input stop.
Min. air gap (armature-pole)	Specify minimum air gap (x_{min}) between armature and solenoid pole. Note that x_{min} must be a positive value.
Magnetic force constant	Specify the constant η for the calculation of magnetic force. Constant η is described in Section 17.2.5.
ELECTRIC CURRENT	
<u>Constant</u>	
Amperage (boost)	Specify the constant magnitude of actuation amperage (boost).
Time function (one switching event)	Specify the magnitude of amperage (boost + holding) as a function of time for one solenoid switching event.

Amperage Table	Specify time in ascending order in the first column. Specify values of amperage in the second column. Intermediate values are bridged by linear interpolation. **Note:** If the Amperage table is active, then each "Switch On" instant (plus Time Delay) implies the start of the Amperage table. Hence this table has to begin with 0 time/angle value and 0 amperage value. The "Switch Off" events do not need to be specified. If the following "Switch Off" event is not defined (empty cell in timing table), amperage table will be active till the next "Switch On" event. **Note:** If the "Switch Off" event is specified, then the amperage is set to 0 at the Switch off instant + Time delay. The rest of the Amperage table (if active) is ignored.
Further Armature (Valve Body) Data	This bar has to be pressed to specify the armature/valve body areas at input/output end and body stop stiffness and damping at input/output end.
Cross-section (under pressure)	
Body area	Specify the cross-sectional areas subjected to pressure.
Cross-sectional area at input end	Specify the cross-sectional area of armature/valve body at input end.
Cross-sectional area at output end	Specify the cross-sectional area of armature/valve body at output end.
Body diameter	Specify the cross-section diameter subjected to pressure.
Cross-section diameter at input end	Specify the cross-section diameter of armature/ valve body at input end.
Cross-section diameter at output end	Specify the cross-section diameter of armature/ valve body at output end.
If the input/output end of the **Armature** body is attached to **Volume** or **Hydraulic Boundary** element, the **Armature** input/output area will be under pressure. This will produce hydraulic forces which will enter the dynamic equation of motion of **Armature** as external forces. Furthermore, **Armature** velocity multiplied by input/output area will produce flow rate which will enter the continuity equation of the **Volume** element connected to the input/output end (refer to Sections 8.1.6 and 8.2.5).	
Body stop data	
(Linear or Pseudo-linear Stop model)	
Body stop stiffness at input end	Specify the stiffness of armature/valve body on input end (k_{in_st}). Stiffness is active when piston lift $x < 0$.
Body stop stiffness at output end	Specify the stiffness of armature/valve body on output end (k_{out_st}). Stiffness is active when piston lift $x > X_{lift}$.
Body stop damping at input end	Specify the damping of armature/valve body on input end (c_{in_st}). Damping is active when piston lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Body stop damping at output end	Specify the damping on of armature/valve body output end (c_{out_st}). Damping is active when piston lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).

(Nonlinear Stop model)	
Body stop linear stiffness at input end	Specify the linear stiffness of armature/valve body on input end (k_{in_st}). Stiffness is active when piston lift $x < 0$.
Body stop linear stiffness at output end	Specify the linear stiffness of armature/valve body on output end (k_{out_st}). Stiffness is active when piston lift $x > X_{lift}$.
Body stop Stiffness factor at input end	Specify the stiffness factor of armature/valve body on input end (n_{in_st}).
Body stop Stiffness factor at output end	Specify the stiffness factor of armature/valve body on output end (n_{out_st}).
Body stop Elastic impact coef. at input end	Specify the elastic impact coefficient of armature/valve body on input end (α_{in_st}).
Body stop Elastic impact coef. at output end	Specify the elastic impact coefficient of armature/valve body on output end (α_{out_st}).
STOP MODEL	
Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (refer to **Figure 54**).
Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function. The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and the ending of contact (refer to **Figure 55**).
Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model is used.
MAGNETIC FORCE	
<u>Program-calculated</u>	
Magnetic Force Calculation Parameters	Press this bar to specify the data necessary for magnetic force calculation.



Dimension of magnetic force	Select the dimension for magnetic force.
Dimension of air gap	Select the dimension for air gap.
Magnetic force constant	Specify the magnetic force constant.
Residual induction constant	Specify the residual induction constant.
Exponent for amperage	Specify the exponent for amperage.
Exponent for air gap	Specify the exponent for air gap.
Force Correction Factor	Press this bar to specify the magnetic force correction factor for one switching event. It can be used to account for magnetic force change due to the variation of actuation amperage and nonlinear magnetic fields effects (e.g. hysteresis).
Switch On	
Force Correction Factor Table	Specify the magnetic force correction factor as a function of Time/Crank Angle. Time t=0 (first table row) implies a Switch On instant.
Switch Off	
Force Correction Factor Table	Specify the magnetic force correction factor as a function of Time/Crank Angle. Time t=0 (first table row) implies a Switch Off instant.
User-defined (3D-table)	
Magnetic Force vs. Lift and Amperage	Press this bar to specify the magnetic force as function of armature lift and amperage in the form of 3D table.
SOLENOID TIMING	
Initial state:	Depending on the initial state different column order in Time/Crank Angle table will appear.
Switched off	
Switched on	
Timing data:	Amperage switching events may be defined in absolute or relative Time/Crank Angle domain.

Absolute (T)	Amperage switching events are defined in absolute time scale in the Time/Crank Angle (CA) table.
Relative (delta T)	Amperage switching events are defined in relative time scale in the Time/Crank Angle (CA) table, i.e. each new switching event refers to the previous event.
Reference:	
Calculation Start	Amperage switching events are referenced to Calculation start.
BTDC (firing)	Amperage switching events are referenced to firing BTDC (time/angle Before Top Dead Center of firing). Active for Absolute Timing Data only.
Time/Crank Angle Table	Specify Time/Crank Angle instants at which amperage is switched on and off.
Time Delay till amperage action	
Delay from Switch ON	Specify the time delay from Switch On instant till amperage initiation.
Delay from Switch OFF	Specify the time delay from Switch Off instant till Amperage initiation.
DIRECTION OF ELEMENT MOTION	
x(e1) **y(e2)** **z(e3)**	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and magnetic force (F_{magn}).	

17.2.2. Initial Conditions

Initial Conditions of **Armature (Extended model)** are identical to the **Initial Conditions** of **Armature (Basic model)** (refer to Section 17.1.2).

17.2.3. Modify Parameter

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

cross-sectional area at input end	SI Unit:	m^2
cross-sectional area at output end	SI Unit:	m^2
Coulomb friction force	SI Unit:	N
armature stroke (maximum lift)	SI Unit:	m
magnetic force constant	SI Unit:	$N*m^2/A^2$
magnitude of Amperage	SI Unit:	A
cross-section diameter at input end	SI Unit:	m
cross-section diameter at output end	SI Unit:	m

17.2.4. Output Parameters

Path: **Element | Store Results**

The **Output Parameters Dialog Box** of **Armature (Extended model)** is shown in Figure 148.

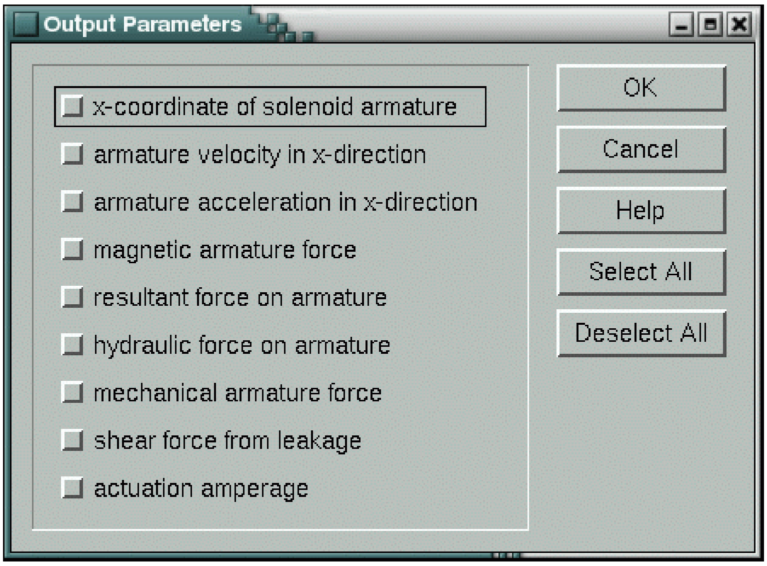


Figure 148: Output Parameters of Armature (Extended model) Dialog

To activate output parameter, the check button in front of output parameter has to be checked.

Description of output parameters:

armature acceleration in x-direction	Armature coordinate and velocity are calculated according to Equation 291 as given in Section 17.2.5.
magnetic force	Magnetic force is calculated from the Equation 291 in accordance with the solenoid timing.

17.2.5. Equation of Motion

Motion of **Armature** in local coordinate system is governed by the following equation:

$$
\begin{aligned}
& m\ddot{x} \\
& + \sum_{i=1}^{n} c_i \left[\dot{x}\cos(\alpha - \alpha_{mc\_i}) - \dot{x}_i \cos(\alpha_{mc\_i} - \alpha_i) - \dot{y}_i \sin(\alpha_{mc\_i} - \alpha_i)\right]\cos(\alpha - \alpha_{mc\_i}) \\
& + \sum_{i=1}^{n} k_i \left[x\cos(\alpha - \alpha_{mc\_i}) - x_i \cos(\alpha_{mc\_i} - \alpha_i) - y_i \sin(\alpha_{mc\_i} - \alpha_i)\right]\cos(\alpha - \alpha_{mc\_i}) \\
& - \sum_{j=1}^{m} c_j \left[\dot{x}\cos(\alpha_{mcj} - \alpha) - \dot{x}_j \cos(\alpha_j - \alpha_{mc\_j}) + \dot{y}_j \sin(\alpha_j - \alpha_{mc\_j})\right]\cos(\alpha_{mcj} - \alpha), \\
& - \sum_{j=1}^{m} k_j \left[x\cos(\alpha_{mcj} - \alpha) - x_j \cos(\alpha_j - \alpha_{mc\_j}) + y_j \sin(\alpha_j - \alpha_{mc\_j})\right]\cos(\alpha_{mcj} - \alpha) \\
& - \sum_{i=1}^{n} F_{0i} \cos(\alpha - \alpha_{mc\_i}) + \sum_{j=1}^{m} F_{0j} \cos(\alpha_{mc\_j} - \alpha) \\
& = F_{magn}(x) - F_{hyd} - F_{frict} - F_{in\_st} - F_{out\_st}
\end{aligned}
\qquad (291)
$$

where:

m	armature mass
x	local x-coordinate of armature
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends
Forces on right side of equation are defined in local x-direction:	
$F_{magn}(x)$	lift-dependent magnetic force

F_{hyd}	hydraulic force
F_{in_st} and F_{out_st}	Additional forces from input and output stops
F_{frict}	Coulomb friction force

Hydraulic force F_{hyd} is calculated from the same Equation 287 as for the **Armature (Basic model)**.

Magnetic force F_{magn} for a stationary state can be calculated from the equation:

$$F_{magn}(x_{gap}) = \frac{1}{2} I^2 \frac{dL}{dx_{gap}}, \tag{292}$$

where I is the magnitude of the actuation current (amperage), L is the inductivity of the coil and x_{gap} is the actual gap between the armature guide and solenoid pole.

Inductivity L is given by:

$$L = \frac{N\Phi}{I} = \frac{N\Theta}{IR_m}, \tag{293}$$

where N is the number of windings (turns), Φ, Θ and R_m are flux, penetration and resistance of the magnetic field, respectively.

Magnetic resistance R_m can be expressed as follows:

$$R_m = \frac{x_{gap}}{\mu A_q}, \tag{294}$$

where μ is the permeability and A_q is the cross-sectional area of the magnetic flux (total active area of poles). Permeability μ is a product of the two constants:

$$\mu = \mu_r \mu_0, \tag{295}$$

where μ_r is the material permeability number (for air $\mu_r \approx 1$) and $\mu_0 = 4\pi 10^{-7}\,\mathrm{H/m}$ is the magnetic field constant.

Magnetic penetration is equal to the amperage multiplied by the number of windings: $\Theta = NI$. Thus, inductivity L is governed by the equation:

$$L = \frac{N^2 I}{IR_m} = \frac{N^2 \mu A_q}{x_{gap}}. \tag{296}$$

Substituting equation (296) into the equation (292), we obtain the equation for the magnetic force:

$$F_{magn}(x_{gap}) = -\frac{1}{2} I^2 \frac{N^2 \mu A_q}{x_{gap}^2} = -\frac{\eta I^2}{x_{gap}^2}, \tag{297}$$

where η we denote as magnetic force constant given by

$$\eta = \frac{1}{2} L x_{gap} = \frac{1}{2} N^2 \mu A_q \,. \tag{298}$$

Magnetic force constant η is the property of the electromagnet.

In real electromagnets, magnetic force often does not follow the square proportionality (297). Usually, the dependence of magnetic force on the amperage and gap is much more flat, i.e. only slightly nonlinear. Moreover, equation (298) is applicable only for a stationary case.

For non-stationary state (switch on/off) additional nonlinear effects like residual induction (hysteresis) occur. To account for these phenomena, equation (297) is considered in a more general form

$$F_{magn}(x_{gap}) = \frac{\eta I^{\alpha}}{x_{gap}^{\beta}} + \frac{\eta_0}{x_{gap}^{\beta}} \,, \tag{299}$$

where α and β are exponents and η_0 is the „constant“ of residual induction caused by remanence magnetic field (at $I = 0$). Default values are α=β =2 and η_0=0, so that equation (299) reduces to the classical equation (297).

Amperage $I = I(t)$ is typically a time function consisting of the boost (pick-up) current and holding current. Boost current is applied at the switching on of the solenoid (maximum air gap) and usually has as a constant duration. After the armature reaches its maximum lift, the boost current can be reduced to the holding current. It has to be sufficient to hold the armature at the top open position.

Air gap x_{gap} is calculated from the formula:

$$x_{gap} = x_{min} + X_{lift} - x \,, \tag{300}$$

where X_{lift} is the armature stroke (distance between input and output stops) , x is the armature lift and x_{min} is the minimum air gap between the armature guide and solenoid pole (gap at $x = X_{lift}$). It should be emphasized that the armature lift has its origin (zero position x=0) at the input stop (valve seat).

Alternatively to the theoretical model, maximum magnetic force $F_{\max}$ can be specified by the user as a function of the armature lift and amperage. The actual magnetic force is then calculated by:

- for Switch ON event:

$$F_{magn}(x) = k_{ON}(t - t_{ON}) F_{max}(x, I) \,, \tag{301}$$

- for Switch OFF event:

$$F_{magn}(t_{OFF}) = k_{ON}(t_{OFF}) F_{max}(x, I) \,,$$

$$F_{magn}(t) = k_{OFF}(t - t_{OFF}) F_{magn}(t_{OFF}) \,, \tag{302}$$

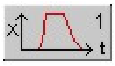
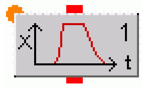
where $k(t)$ is the force correction factor ($0 \le k(t) \le 1$) and I is actuation amperage. Force correction factor enables users to take into account additional losses and other transient effects.

The maximum value of magnetic force at each air gap value is directly picked from the 3-dimesional magnetic force table. Intermediate values are calculated by linear interpolation. Note that no extrapolation is performed, i.e. any air gap and amperage value outside of the table range is invalid.

Forces from input and output stops F_{in_st} and F_{out_st} are calculated in the same way as for the **Standard Piston**.

18. PIEZO

18.1. Lift Function

Element Name:	**Lift Function**	
Element Icon:		
Definition:	This element serves to define 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 piezo stack end.	
Connecting pins:	standard pins: 2 (mechanical) special pins: 0 wire pins: 1	

> **Note:** Piezo Lift Function is one-dimensional with a possibility to define the motion in x-direction or y-direction.

Piezoelectric stack actuator is an assembly of thin layers (wafers) of electrically active ceramics. The wafers are contacted in parallel (refer to Figure 149).

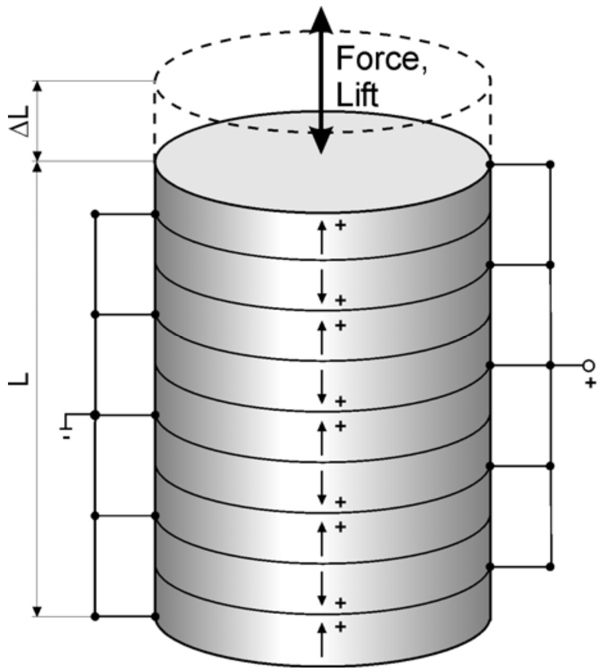


Figure 149: Piezoelectric Stack

18.1.1. Input Parameters

Description of input data for the **Lift Function**:

Element name	The name of the element is specified as default.
Maximum body lift	Specify the maximum lift of Piezo stack end X_{lift}.
Direction: x/y-axis (flow)	Switches between x-axis and y-axis direction. Note that the only hydraulic flow is always directed along the x-axis. Program will calculate Piezo stack coordinate only in the selected direction.
Charge duration	Specify the charging duration of Piezo stack.
Discharge duration	Specify the discharging duration of Piezo stack.
ACTUATOR TIMING	
Timing data:	
Absolute (T)	Charging/discharging events are defined in absolute time scale in the Time/Crank Angle (CA) table.
Relative (delta T)	Charging/discharging events are defined in relative time scale in the Time/Crank Angle (CA) table, i.e. each new event refers to the previous event.
Reference:	
Calculation Start	Charging/discharging events are referenced to Calculation start.
BTDC (firing)	Charging/discharging events are referenced to firing BTDC (time/angle Before Top Dead Center of firing). Active for Absolute Timing Data only.
Time/Crank Angle (CA) Table	Specify Time/Crank Angle instants at which Piezo stack charging and discharging begins (maximum 110 pairs). Piezo stack charging and discharging processes cannot interfere with each other, i.e. discharging can begin only when charging is over and vice versa.

18.1.2. Initial Conditions

Initial conditions cannot be specified for **Lift Function** element.

18.1.3. Modify Parameter

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

maximum piezo lift	SI Unit:	m
charging duration	SI Unit:	s
discharging duration	SI Unit:	s

18.1.4. Output Parameters

Path: **Element│ Store Results**

To activate output parameter, the option box in front of output parameter has to be checked.

Description of output parameters:

x-coordinate	Nonzero only if x-direction is specified in **Input** Dialog Box.
y-coordinate	Nonzero only if y-direction is specified in **Input** Dialog Box.

Coordinate of **Piezo Lift Function** is calculated from Equation (303).

18.1.5. Equation of Motion

Formulas for calculation of coordinate according to actual time are given by:
(same procedure for both x and y directions):

$$t < t^i_{start} \qquad x(t) = 0, \qquad i = 1........9$$

$$t^i_{start} < t < t^i_{start} + T_{start} \quad x(t) = \frac{1}{2} X_{lift} \left[1 - cos \left(\frac{\pi \left(t - t^2_{start} \right)}{T_{start}} \right) \right], \tag{303}$$

$$t^i_{start} + T_{start} < t < t^i_{end} \qquad x(t) = X_{lift},$$

$$t^i_{end} < t < t^i_{end} + T_{end} \quad x(t) = \frac{1}{2} X_{lift} \left[1 - cos \left(\frac{\pi \left(T_{end} + t - t^2_{end} \right)}{T_{end}} \right) \right],$$

$$t^i_{end} + T < t < t^{i+1}_{start} \qquad x(t) = 0.$$

where:

x	coordinate of piezo stack
t	actual time
X_{lift}	maximum lift of piezo stack
$T_{star\,t}$	charge duration of Piezo stack
T_{end}	discharge duration of Piezo stack
t^i_{start}	start time of i^{th} charging event
t^i_{end}	start time of i^{th} discharging event

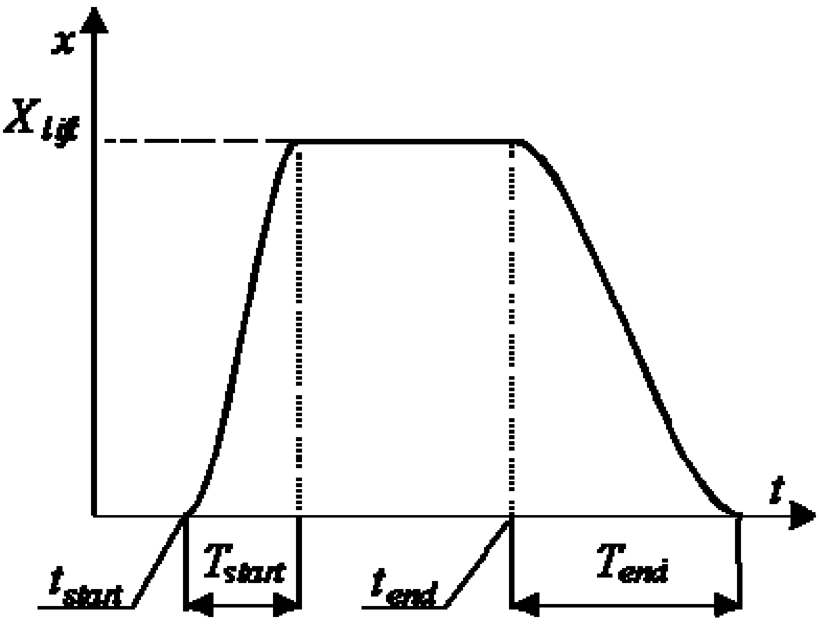
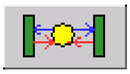
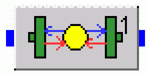


Figure 150: Piezo Stack Actuator Lift

18.2. Lift Amplifier

Element Name:	**Amplifier**	
Element Icon:		
Definition:	This element serves to define a lift amplifier (converter). It converts the motion of two connected pistons into the forces acting on these pistons. Basically it is a two-dimensional force-displacement function.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 0 Only one input and one output hydraulic connection has to be specified.	

Note: Currently the Variable-step solver does not support this element.

☞

Note: The **Lift Amplifier** calculates control forces from positions of connected piston-type elements on input and output sides. Positive direction of forces is shown in Figure 152.

Note: **Lift Amplifier** element can be connected with **Piston**, **Plunger** and **Split-Injection Piston** elements. **Lift Amplifier** must always be connected to the two piston-type elements – one on input and one on output side.

Note: Connections must formally be hydraulic because information about the force from **Lift Amplifier** to piston-type element is transferred via pressure. It means that the respective **Piston** must have defined area/diameter on its end connected to **Lift Amplifier**. The size of area is not important because **Lift Amplifier** element will adjust pressure to transmit the specified force to piston. This force will act via pressure as a hydraulic connection force.

Lift Amplifier is typically used to model the amplification of Piezo stack motion through a force-displacement function. Different types of piezoelectric valves with piezo motion amplifiers are shown in Figure 151.

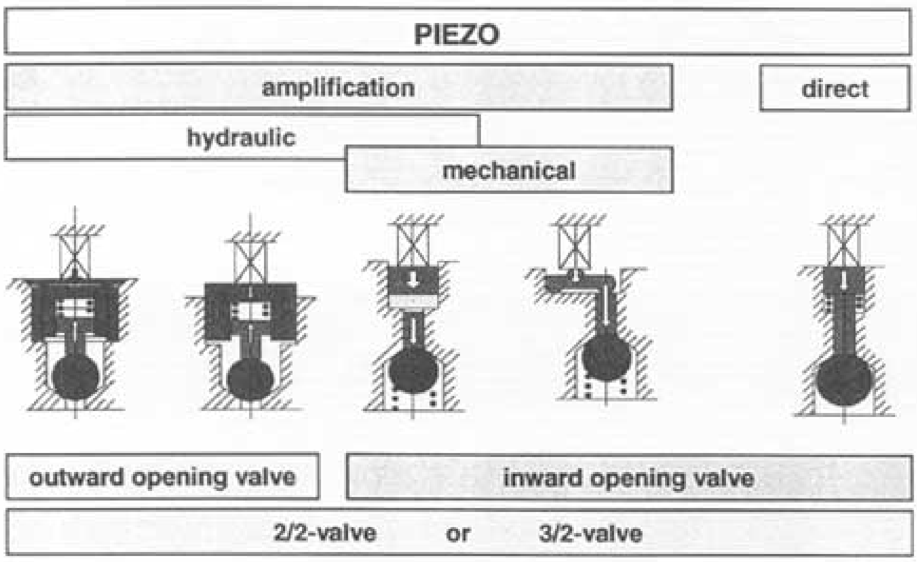


Figure 151: Piezoelectric Valves with Piezo Lift Amplification

18.2.1. Input Parameters

Description of input data for the **Lift Amplifier**:

Element name	The name of the element is specified as default.
INPUT SIDE (F1)	
PISTON DISPLACEMENT	
Number of positions S1 (input)	Number of columns in table (positions of input piston). Maximum number of columns in table is 10.
Number of positions S2 (output)	Number of rows in table (positions of output piston). Maximum number of rows in table is 20. Additional rows (if specified) will be ignored.
Starting value S1 (input)	Starting position (initial lift) of piston connected to input end.
Starting value S2 (output)	Starting position (initial lift) of piston connected to output end.
Increment S1 (input)	Increment of position for piston connected to input end.
Increment S2 (output)	Increment of position for piston connected to output end.
CONTROL FORCE	
	Specify the input control force according to positions of piston-type elements connected to input and output ends. S1(i) lift of the input piston (max. 10 columns) S2(j) lift of the output piston (max. 20 rows) Each force value in the table corresponds to the specific positions of input and output pistons-type elements. Intermediate values are calculated by linear interpolation. Force values outside the defined range of table arguments (piston positions) are undefined. If such a situation occurs, an error message will be produced.
OUTPUT SIDE (F2)	**TAB**
PISTON DISPLACEMENT	
Number of positions S1 (input)	Number of columns in table (positions of input piston). Maximum number of columns in table is 10.
Number of positions S2 (output)	Number of rows in table (positions of output piston). Maximum number of rows in table is 20.
Starting value S1 (input)	Starting position (initial lift) of piston connected to input end.
Starting value S2 (output)	Starting position (initial lift) of piston connected to output end.
Increment S1 (output)	Increment of position for piston connected to input end.
Increment S2 (output)	Increment of position for piston connected to output end.

CONTROL FORCE	
	Specify the output control force according to positions of piston-type elements connected to input and output ends. S1(i) lift of the input piston (max. 10 columns) S2(j) lift of the output piston (max. 20 rows) Each force value in the table corresponds to the specific positions of input and output piston-type elements. Refer to the description of Control Force of Input Side (F1) for information on entering data into the table.

18.2.2. Initial Conditions

Initial conditions cannot be specified for **Lift Amplifier** element.

18.2.3. Modify Parameter

Modifiable parameters cannot be specified to **Lift Amplifier** element.

18.2.4. Output Parameters

Output Parameters cannot be specified for **Lift Amplifier** element.

18.2.5. Basic Relationships

Lift Amplifier calculates the control forces acting on the connected piston-type elements depending on their positions. Positive direction of the control forces is shown Figure 152. These forces can be defined as follows:

$$F_{in} = F_{in}(x_{in}, x_{out}) \quad (304)$$

$$F_{out} = F_{out}(x_{in}, x_{out})$$

where F_{in} and F_{out} are the control forces imposed on input and output pistons and x_{in} and x_{out} are the coordinates of input and output pistons, respectively.

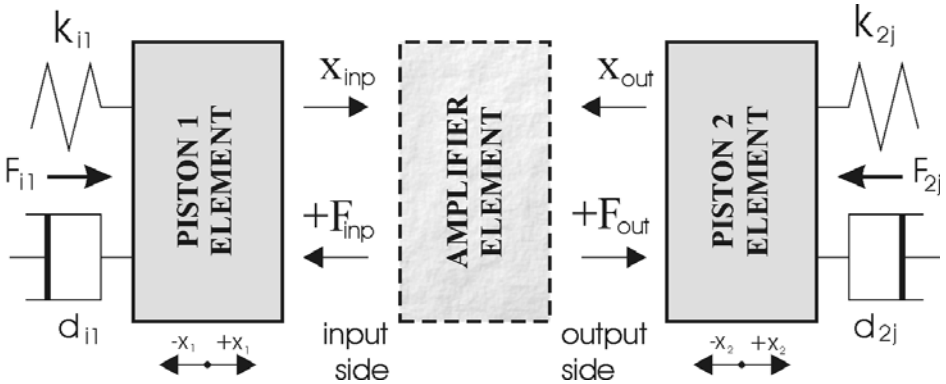


Figure 152: Lift Amplifier element with Two Pistons

Two-dimensional force-displacement function in Equation 304. is discretized: forces F_{in} and F_{out} (in separate tables) are specified only at grid points. For intermediate lift values x_{in} and x_{out}, actual force value is obtained by interpolation according to four neighboring grid points:

$$F_{in}(x_{in}, x_{out}) = \{(x - x_{in}(i-1))(x_{out} - x_{out}(j-1))\, F_{in}(x_{in}(i), x_{out}(j))\}$$

$$-\{(x_{in} - x_{in}(i-1))(x_{out} - x_{out}(j))\, F_{in}(x_{in}(i), x_{out}(j-1))\} \tag{305}$$

$$-\{(x_{in} - x_{in}(i))(x_{out} - x_{out}(j-1))\, F_{in}(x_{in}(i-1), x_{out}(j))\}$$

$$+\{(x_{in} - x_{in}(i))(x_{out} - x_{out}(j))\, F_{in}(x_{in}(i-1), x_{out}(j-1))\} / \Delta x_{in}\, \Delta x_{out},$$

$$F_{out}(x_{in}, x_{out}) = \{(x_{in} - x_{in}(i-1))(x_{out} - x_{out}(j-1))\, F_{out}(x_{in}(i), x_{out}(j))\}$$

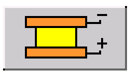
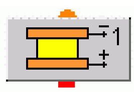
$$-\{(x_{in} - x_{in}(i-1))(x_{out} - x_{out}(j))\, F_{out}(x_{in}(i), x_{out}(j-1))\}$$

$$-\{(x_{in} - x_{in}(i))(x_{out} - x_{out}(j-1))\, F_{out}(x_{in}(i-1), x_{out}(j))\} \tag{306}$$

$$+\{(x_{in} - x_{in}(i))(x_{out} - x_{out}(j))\, F_{out}(x_{in}(i-1), x_{out}(j-1))\} / \Delta x_{in}\, \Delta x_{out}$$

where Δx_{in} and Δx_{out} are position increments of input and output pistons, respectively.

18.3. Stack Actuator

Element Name:	**Stack Actuator**	
Element Icon:		
Definition:	This element serves to define a Piezoelectric Stack Actuator. It is a lumped-parameter model with rate-independent hysteresis describing the accurate actuator behavior in electrical and mechanical domains.	
Connecting pins:	standard pins: 1 (mechanical) special pins: 0 wire pins: 1	

☞ **Note:** Currently the Variable-step solver does not support this element.

☞

Note: **Stack Actuator** is one-dimensional, elastic, multi-layer element with the piezoelectric stack motion in local x-direction only.

Note: **Stack Actuator** element can be connected to a **Piston**, **Plunger** or **Needle** element only. The connection stiffness usually has to be very high to preserve the accurate **Actuator** motion.

Note: Due to a very small displacement and high connection stiffness **Stack Actuator** is integrated with a smaller time step than the rest of the system.

Note: For **Stack Actuator** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: Local x-axis of **Stack Actuator** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter: Direction of Element motion.

Piezoelectric stack actuator consists of a dense pile of separately contacted layers (wafers) of electrically active ceramic material. These wafers are connected in parallel as shown in Figure 149. Piezoelectric stack typically has circular or rectangular cross-section.

18.3.1. Input Parameters

Description of input data for the **Piezo Actuator**:

Element name	The name of the element is specified as default.
DATA OF ONE CERAMIC LAYER	
Layer mass	Specify the mass (m_1) of one ceramic layer.
Mechanical stiffness	Specify the stiffness of one ceramic layer (k_1).
Mechanical damping	Specify the damping of one ceramic layer (b_1).
Electric capacitance	Specify electrical capacitance of one ceramic layer (C_1).
Piezo Gain	
Constant	
Gain (d33)	Specify the piezoelectric gain of one ceramic layer (d_{33}).
Variable	
Function of Voltage	Press this bar to specify the piezo gain as voltage function in table form.

Piezo Strain along Polarization (d33)	Specify voltage in ascending order in the first column. **Note:** Voltage values in the first column are independent of the Input voltage. Specify piezo gain values in the second column. For voltage values higher than the last value in the table, the gain value from the last table row is used. For voltage values lower than the first value in the table, the gain value from the first table row is used. No gain extrapolation outside of table range is performed.
Transformer ratio	
Electromechanical transformer ratio	Specify the electromechanical transformer ratio (*n*).
INPUT VOLTAGE	
Reference (Time/CA):	
Calculation Start	Time/Crank Angle column is referenced to Calculation start.
BTDC (firing)	Time/Crank Angle column is referenced to firing BTDC (time/angle Before Top Dead Center of firing).
Input voltage table	Specify time/crank angle values in ascending order in the first column. Specify voltage values in the second column (V_{inp}). For time/crank angle values higher than the last value in the table, the voltage value from the last table row is used. For time/crank angle values lower than the first value in the table, the voltage value from the first table row is used. No voltage extrapolation outside of table range is performed. **Note**: If Normalized Stroke–Voltage diagram is active (refer to Figure 156), Input Voltage table has to be specified within the voltage range in the Stroke-Voltage diagram.
Number of ceramics layers	Specify total number of ceramic layers (N_L) in the stack.
Preload force	Specify constant preload force on the piezo stack.
Mechanical transmission ratio	Specify the mechanical transmission ratio (if any) between the Stack Actuator and connected element.
HYSTERESIS (of one ceramic layer)	
Maxwell resistive Capacitor (MRC) elements	
Precharging voltage	Specify maximum precharging voltage (if precharging has to be applied).

Normalized Stroke-Voltage diagram (idle)	
Max. idle stroke of one layer	Specify maximum idle stroke for one ceramic layer from normalized Stroke-Voltage table (X_{max}).
MRC element table	Specify electrical stiffness in the first column. Specify break voltage (electromotive force) values the in second column. **Note**: Maxwell Resistive Capacitor (MRC) elements are used to describe the hysteretic behavior of piezoelectric ceramics (refer to Equations 310 and 311). At least one Maxwell element is required.
Stroke-Voltage table	Specify voltage values in the first column. Specify relative (normalized) idle stroke values in the second column. **Note**: Normalized Stroke–Voltage diagram describes the hysteresis behavior of one ceramic layer. It has to be defined for idle piezoelectric stack because it is usually available in this form only (no external load on the stack). Typically, the Stroke–Voltage diagram has to be specified by three branches (refer to Figure 156): initial charging branch (lower curve), discharging branch (upper curve) and recharging branch (middle curve). Note that start voltage has to be smaller or equal to the initial input voltage, minimum voltage can be negative (for bipolar operation) and maximum voltage must be positive. **Note**: In the current version Stroke-Voltage diagram yields accurate results for the idle case only. For small external loads sufficient accuracy can be expected. For the higher loads piezo stack stroke can get distorted.
DIRECTION OF ELEMENT MOTION	
x(e1) **y(e2)** **z(e3)**	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of input and output stop forces (F_{in_st}, F_{out_st}) and magnetic force (F_{magn}).	

18.3.2. Initial Conditions

Initial conditions cannot be specified for **Stack Actuator** element.

18.3.3. Modify Parameter

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

stiffness of ceramic layer	SI Unit:	N/m
damping of ceramic layer	SI Unit:	Ns/m
electric capacitance of layer	SI Unit:	F
electromechanical transformer ratio	SI Unit:	C/m
preload force	SI Unit:	N
mechanical transmission ratio	SI Unit:	-

18.3.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option box in front of output parameter has to be checked.

Description of output parameters:

stroke of piezo actuator (with transmission)	Displacement of Piezo Stack Actuator relative to the connected element (precharging, preload force and mechanical ratio are included).
stroke of piezoelectric stack alone	End point displacement of Stack Actuator (x_n) without mechanical ratio.
piezo actuator velocity (with transmission)	Velocity of Stack Actuator in x-direction ($\dot{x}_n$).
piezo actuator acceleration	Acceleration of Stack Actuator in x-direction ($\ddot{x}_n$).
electric charge in piezoelectric stack	Electric charge in the piezoelectric ceramics (q).
transduced force of piezoelectric stack	Electromechanical force generated by the piezoelectric stack (F_t).
resultant force on piezo stack actuator	Resultant force on piezo stack actuator is the sum of all connections and external forces acting on it. $$F_R = m\ddot{x}_n$$ (refer to Section 18.3.5)

driving power of piezo stack actuator	Driving power is a product of transduced force and stack actuator velocity: $$P = F_t \cdot \dot{x}_n$$
static idle stroke of piezoelectric stack	Static idle stroke of piezoelectric stack is given by: $$X_0 = N_L \cdot \frac{n}{k_1} \cdot V_t ,$$ where V_t is the back electromotive force from the mechanical domain. **Note**: For convenience, static idle stroke is calculated at each calculation step, formally treating the actual input voltage as a static voltage.
static blocking force of piezoelectric stack	Static blocking force is maximum force generated by piezoelectric stack blocked by an infinitely rigid restraint: $$F_B = k_n \cdot X_0 ,$$ where k_n is the total mechanical stiffness of piezoelectric stack: $$k_n = \frac{k_1}{N_L}$$
input voltage on piezoelectric stack	Input voltage applied on piezoelectric stack is: $$V_{inp} = V_t + V_{rc} ,$$ where V_{rc} is the voltage loss due to hysteresis.

18.3.5. Equation of Motion

Piezoelectric **Stack Actuator** model is based on the rate-independent hysteresis exhibited between the voltage and displacement[29]. This hysteretic behavior can be defined either by the Maxwell Resistive Capacitor model (refer to Figure 153) or Stroke-Voltage diagram (refer to Figure 156).

Maxwell Resistive Capacitor (MRC) model

Hysteretic (stick-slip) behavior of the piezoelectric stack containing n elasto-slide elements can be described by the equation:

$$F_i = \begin{cases} k_i \cdot (x - x_{b_i}) & if \quad \left| k_i \cdot (x - x_{b_i}) \right| < f_i \\ f_i \cdot \mathrm{sgn}(\dot{x}) & and \quad x_{b_i} = x - \frac{f_i}{k_i} \cdot \mathrm{sgn}(\dot{x}) \quad else \end{cases} \tag{307}$$

[29] Goldfarb M. and Celanovic N. *A Lumped Parameter Electromechanical Model for Describing the Nonlinear Behavior of Piezoelectric Actuators*. Journal of Dynamic System, Measurement and Control, 1997, Vol. 119, pp. 478-485.

$$F = \sum_{i=1}^{n} F_i \tag{308}$$

where:

x	input displacement
F	output force
F_i	output force of the i-th elasto-slide element
k_i	spring stiffness of the i-th elasto-slide element
f_i	breakaway force of the i-th elasto-slide element
x_{bi}	block position of the i-th elasto-slide element

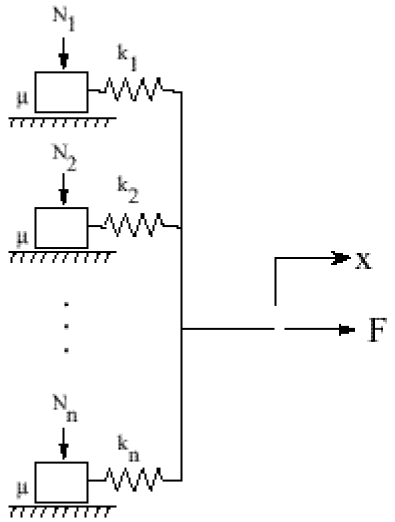


Figure 153: Equivalent System of Elasto-slide Elements

Figure 154 shows the hysteretic force-displacement behavior of the system in Figure 153.

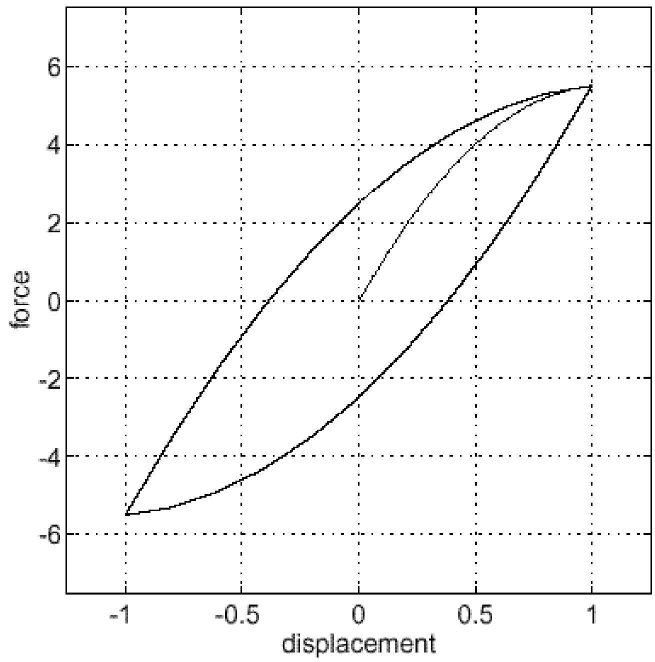


Figure 154: Force-displacement Behavior of Elasto-slide Elements

The energy-based constitutive relationships of the Maxwell model are not domain specific, and can therefore represent any rate-independent hysteretic relationship between a generalized force and generalized displacement in a lumped parameter casual form. Hence generalized Maxwell model can represent rate-independent hysteresis between voltage (generalized force) and charge (generalized displacement) in Maxwell Resistive Capacitor element (refer to Equations 310 and 311).

The dynamic equations of motion of the piezoelectric stack are given by (refer to Figure 155):

$$\begin{aligned}
&q = n \cdot x_n + C_n \cdot V_t, \\
&V_t = V_{in} - V_{rc}, \\
&V_{rc} = mrc(q), \\
&F_t = n \cdot V_t, \\
&m_n \cdot \ddot{x}_n + b_n \cdot \dot{x}_n + k_n \cdot x_n = F_t + F_{ext},
\end{aligned} \tag{309}$$

where:

$$m_n = \frac{N_L + 1}{2} \cdot m_1$$

$$b_n = \frac{b_1}{N_L}$$

$$k_n = \frac{k_1}{N_L}$$

$$C_n = C_1 \cdot N_L$$

q	total charge in the piezoelectric stack
n	electromechanical transformer ratio
x_n	endpoint displacement of the piezoelectric stack
C_n	electrical capacitance of the piezoelectric stack
V_{in}	input voltage applied on the piezoelectric stack
V_{rc}	voltage across the MRC element (voltage loss due to hysteresis)
V_t	back electromotive force from the mechanical domain
F_t	transduced force from the electrical domain
m_n	equivalent mass of the piezoelectric stack
b_n	damping of the piezoelectric stack
k_n	stiffness of the piezoelectric stack
N_L	number of ceramic layers
F_{ext}	force imposed from the external load
mrc( )	function relating the voltage across the MRC element to the charge in the ceramic (refer to Equations 310 and 311).

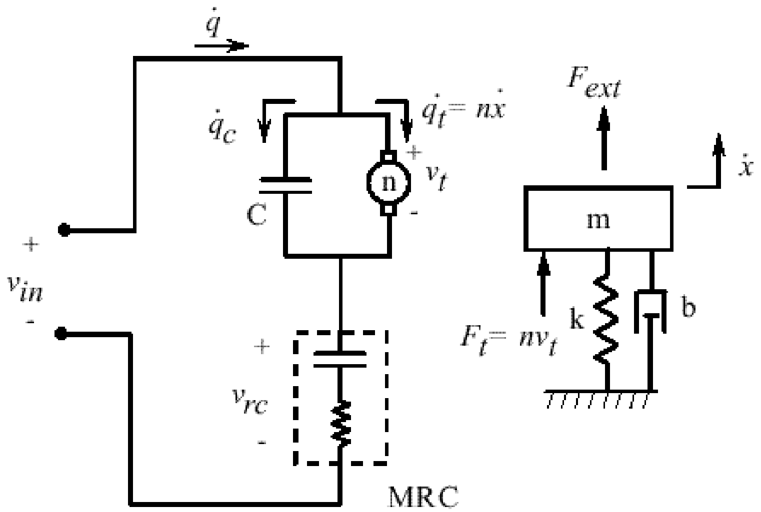


Figure 155: Schematic Representation of Stack Actuator Model

Voltage across the MRC element is calculated from:

$$(V_{rc})_i = \begin{cases} k_i \cdot \left[q - (q_{rc})_i\right] & if \quad \left|k_i \cdot (q - (q_{rc})_i)\right| < f_i \\ \end{cases} \tag{310}$$

$$f_i \cdot sgn(\dot{q}) \qquad and \qquad (q_{rc})_i = q - \frac{f_i}{k_i} \cdot sgn(\dot{q}) \quad else$$

$$V_{rc} = \sum_{i=1}^{n} (V_{rc})_i \tag{311}$$

where:

q	input charge
V_{rc}	output voltage
$(v_{rc})_i$	output voltage of the i-th MRC element
k_i	electric stiffness of the i-th MRC element
f_i	breakaway force of the i-th MRC element
$(q_{rc})_i$	charge of the i-th MRC element

Normalized Stroke-Voltage Diagram

Alternative, more practical way of modeling the hysteresis behavior of the piezoelectric stack is based on the normalized Stroke–Voltage diagram (refer to Figure 156). This diagram is usually available for the idle case only. Therefore the method gives accurate results for idle case or small loads (compared to the transduced force). For higher external loads, piezoelectric stack stroke and other results may become inaccurate.

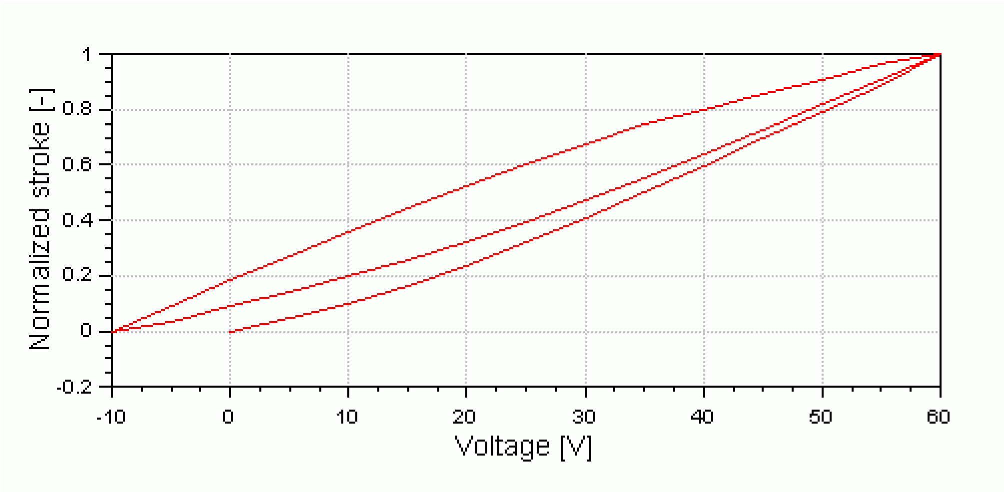


Figure 156: Normalized Stroke–Voltage Diagram

The dynamic equations of the piezoelectric stack motion are the same as for MRC model (refer to Equation 309) except the calculation of electromotive force from the mechanical domain V_t. For idle and load case it is determined from the normalized Stroke–Voltage diagram using the formulae:

$$V_{t0} = \frac{X_a \cdot X_{max} \cdot k_1}{N_L \cdot n} \quad , \tag{312}$$

$$V_t = V_{t0} + \frac{q_{idle} - q_1}{C_1}, \tag{313}$$

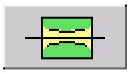
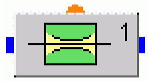
where:

$$q_{idle} = n \cdot X_a \cdot X_{max} + C_1 \cdot V_{t0}.$$

V_{t0}	back electromotive force from the mechanical domain for idle case
V_t	back electromotive force from the mechanical domain for load case
X_a	relative stroke of one ceramic layer from the Stroke-Voltage diagram
X_{max}	absolute value of the maximum idle stroke of one ceramic layer
K_1	stiffness of one ceramic layer
N_L	number of ceramic layers
N	electromechanical transformer ratio
q_{idle}	charge in one ceramic layer for idle case
Q_1	actual charge in one ceramic layer
C_1	electrical capacitance of one ceramic layer

19. ORIFICE

19.1. Standard Orifice

Element Name:	**Standard Orifice**	
Element Icon:		
Definition:	This element serves to define the properties of a Standard Orifice (laminar/turbulent flow model).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Note: Turbulent flow resistance in **Standard Orifice** can be constant or variable (pressure-dependent). Constant flow resistance can be defined by entering the coefficient value or the flow rate and pressure difference across the orifice. In this case the flow resistance coefficient is calculated directly in GUI window.

Note: In combination with the **Modify Parameter** function, **Standard Orifice** can be used as switch-orifice (time-controlled throttle). However, a **Variable Throttle (Flow Area vs. Time/CA)** is more suitable for this purpose.

Schematic of **Standard Orifice** with expansions/contractions at connections is shown in Figure 157.

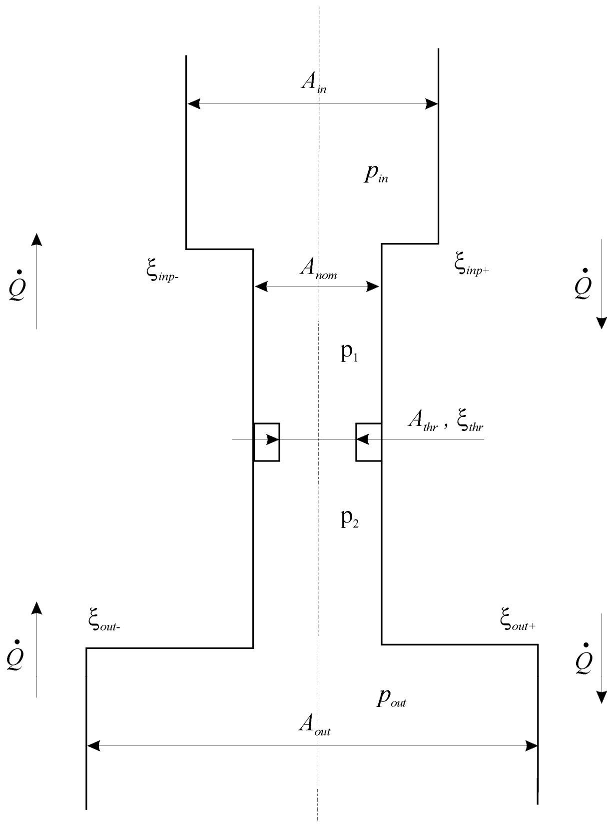


Figure 157: Standard Orifice Geometry

19.1.1. Input Parameters

Description of input data for the **Standard Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
CROSS-SECTION AREA	
Tube cross-sectional area	Specify cross-sectional area of orifice tube A_{nom} (refer to Figure 157)
Throttle cross-sectional area	Specify cross-sectional area of throttle A_{thr} (refer to Figure 157).
CROSS-SECTION DIAMETER	
Tube cross-section diameter	Specify cross-section diameter of orifice tube d_{nom}.
Throttle cross-section diameter	Specify cross-section diameter of throttle d_{thr}.

TURBULENT flow COEFFICIENT in throttle	If this button is active, BOOST Hydsim will use constant flow resistance (ξ thr) or discharge (μ thr) coefficient in the throttle (narrowest area).
Critical flow number (laminar->turbulent)	Specify critical flow number (λ) for transition between laminar and turbulent flow
Resistance coef.	For this option flow resistance coefficient has to be specified.
Flow resistance coefficient (>1)	Specify flow resistance coefficient ξ thr in throttle (narrowest cross-sectional area).
Flow Rate	For this option flow rate through orifice has to be specified. Turbulent flow discharge coefficient is then calculated from Bernoulli equation: $\mu = \frac{Q_m}{A_{thr}}\sqrt{\frac{\rho}{2\Delta p_m}}$ - if throttle cross-section area is activated $\mu = \frac{4Q_m}{\pi d_{thr}^2}\sqrt{\frac{\rho}{2\Delta p_m}}$ - if throttle cross-section diameter is activated
Flow rate through throttle	Specify (measured) flow rate through throttle Qm.
Pressure difference	Specify pressure difference pm at which volumetric flow is measured or obtained otherwise.
Calculate coef.	Press to calculate the flow resistance coefficient and display it at Zhyd position: $Z_{hyd} = \frac{1}{\mu_{hyd}^2}$
Discharge coef.	For this option flow discharge coefficient has to be specified.
Flow discharge coefficient (<1)	Specify flow discharge coefficient μ thr in throttle (narrowest cross-sectional area).
flow resistance in throttle (Variable)	If this button is active, BOOST Hydsim expects variable flow resistance coefficient $\xi_{thr} = f\left(\frac{p_1}{p_2}\right)$ in the throttle.
Variable flow resistance table	Specify the ratio between the input/output (output/input) pressures in ascending order in the first column. Specify flow resistance coefficient in the second column. **Note:** Pressure ratio must be within the range 0…1: $0 < \frac{p_1}{p_2} < 1$. If $p_{in} > p_{out}$, then the program uses the ratio $\frac{p_{out}}{p_{in}}$ and vice versa. Thus, orifice must have a symmetric geometry because input and output flow resistance coefficients are equal.

	Note: Flow resistance coefficient in throttle must be greater equal than 1 : $\xi_{thr} \geq 1$. For pressure ratio higher than the highest value specified, the highest ξ_{thr} value from the table is used. For pressure ratio lower than the lowest value specified, the lowest ξ_{thr} value from the table is taken (kept constant as long as pressure ratio is out of table range).
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS	
Cross-sectional area at input end	Specify cross-sectional area at input end A_{in}. **Note:** If cross-sectional area at input end is zero (empty), it implies $A_{in}= A_{nom}$.
Cross-sectional area at output end	Specify cross-sectional area at output end A_{out}. **Note:** If cross-sectional area at output end is zero (empty), it implies $A_{out}= A_{nom}$.
Positive flow resistance coefficient at input end	Specify resistance coefficient ξ_{int+} for positive flow (from input end to output end) related to A_{nom} at sudden change of cross-sectional area on input end.
Positive flow resistance coefficient at output end	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to A_{nom} at sudden change of cross-sectional area on output end.
Negative flow resistance coefficient at input end	Specify resistance coefficient ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden changes of cross-sectional area on input end.
Negative flow resistance coefficient at output end	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end.

19.1.2. Initial Conditions

Initial conditions cannot be specified for **Standard Orifice** element.

19.1.3. Modifiable Parameters

Path: **Element | Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

tube cross-section area	SI Unit:	m^2
throttle cross-section area	SI Unit:	m^2
flow resistance coefficient in throttle	SI Unit:	---

cross-sectional area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-section area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---
tube cross-section diameter	SI Unit:	m
throttle cross-section diameter	SI Unit:	m

19.1.4. Output Parameters

Path: **Element|Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

static pressure before throttle (input end)	Corresponds to p1 (refer to Figure 157).
static pressure after throttle (output end)	Corresponds to p2 (refer to Figure 157).
volumetric flow rate at orifice input end	To obtain it, fluid flow velocity is calculated by Equation 338 in Section 19.1.6 and multiplied with cross-sectional area at input end Ain.
volumetric flow rate at orifice input end	Output values will be 0 if the Time domain button is active in Calculation Control dialog.
cumulative flow rate at orifice input end	Cumulative flow rate per calculation range.
flow resistance coefficient in throttle	Output values will be constant if constant flow resistance or discharge in throttle is active in input dialog.
volumetric flow rate at orifice output end	To obtain it, fluid flow velocity is calculated by Equation 337 in Section 19.1.6 and multiplied with cross-sectional area at output end Aout.
volumetric flow rate at orifice output end	Output values will be 0 if the Time domain button is active in Calculation Control dialog.
cumulative flow rate at orifice output end	Cumulative flow rate per calculation range.
flow coefficient in throttle	Throttle flow coefficient for laminar and turbulent flow
mean flow velocity in throttle	Average flow velocity of fluid in throttle (narrowest cross-sectional area)
throttle flow number	Flow number value in the throttle
Reynolds number	Reynolds number value in the throttle

19.1.5. Estimation of Flow Resistance Coefficient

In this section, tables and calculation formulas for the estimation of the flow resistance coefficient of two orifice types are given: thick-edged orifice in straight tube and single disk gate valve (both shown in Figure 158).

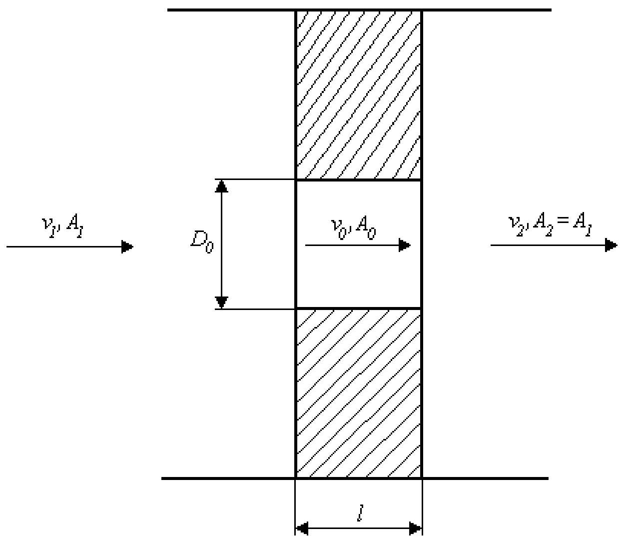


Figure 158: Thick Edged Orifice in Straight Tube

Table 8 contains values of the flow resistance coefficient ξ_{nom} related to the nominal cross-sectional area of orifice. ξ_{nom} values have to be selected according to the ratio between the orifice length and hydraulic diameter l/D_h and ratio between the cross-sectional areas of orifice and tube A_0/A_1. Resistance coefficient can be calculated from the following formula:

$$\xi_{nom} = \left[\left(\frac{1}{2} + \tau\sqrt{1-\frac{A_0}{A_1}}\right)\left(1-\frac{A_0}{A_1}\right) + \left(1-\frac{A_0}{A_1}\right)^2 + \lambda\frac{l}{D_h}\right]\left(\frac{A_1}{A_0}\right)^2 = \left(\xi_0 + \lambda\frac{l}{D_h}\right)\left(\frac{A_1}{A_0}\right)^2,$$

with $$\xi_0 = \frac{1}{2}\left(1-\frac{A_0}{A_1}\right) + \left(1-\frac{A_0}{A_1}\right)^2 + \tau\sqrt{1-\frac{A_0}{A_1}}\left(1-\frac{A_0}{A_1}\right), \tag{314}$$

where λ is the friction factor (function of Reynolds number and surface roughness), D_h is the hydraulic diameter of orifice and $\tau = f(l/D_h)$ is the coefficient accounting for the wall thickness, inlet edge shape and flow passage conditions.

For a circular orifice D_h is equal to orifice diameter D_0 (refer to Figure 158). If orifice has a shape other than a circle, use the following formula to calculate D_h:

$$D_h = \frac{4A_0}{P_0}, \tag{315}$$

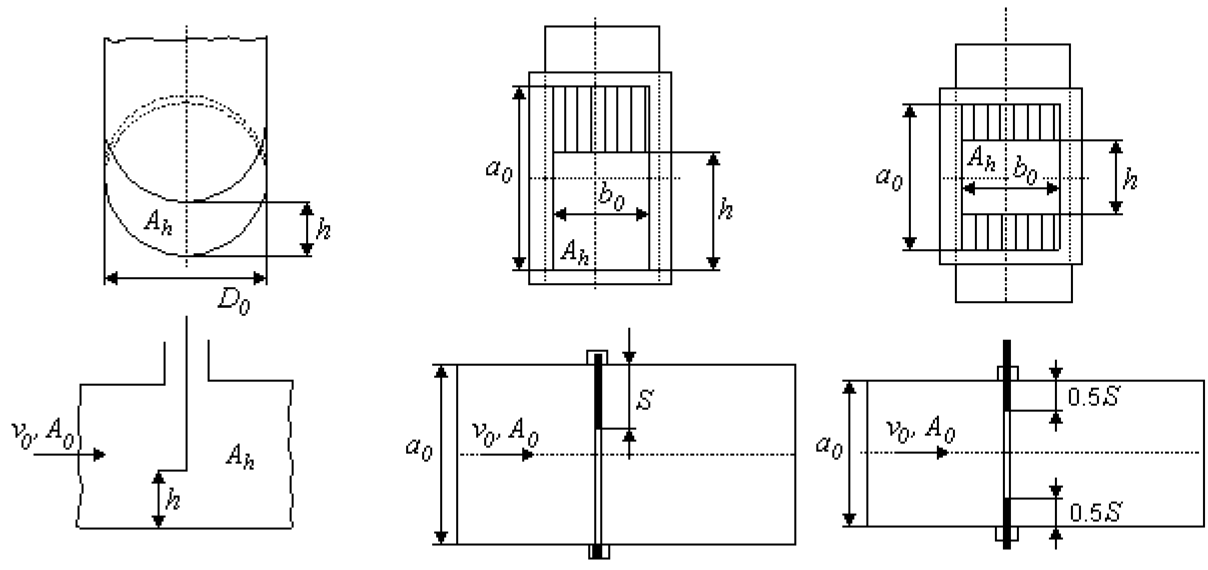
where P_0 is orifice perimeter.

Table 8: Thick-edged Orifice in Straight Tube (λ = 0.02)

ξ_{nom}		A_0/A_1															
l/D_h	τ	0,02	0,04	0,06	0,08	0,10	0,15	0,20	0,25	0,30	0,40	0,50	0,60	0,70	0,80	0,90	1,0
0	1,35	7000	1670	730	400	245	96,0	51,5	30,0	18,2	8,25	4,00	2,00	0,97	0,42	0,13	0
0,2	1,22	6600	1600	687	374	230	94,0	48,0	28,0	17,4	7,70	3,75	1,87	0,91	0,40	0,13	0,01
0,4	1,10	6310	1530	660	356	221	89,0	46,0	26,5	16,6	7,40	3,60	1,80	0,88	0,39	0,13	0,01
0,6	0,84	5700	1380	590	322	199	81,0	42,0	24,0	15,0	6,60	3,20	1,60	0,80	0,36	0,12	0,01
0,8	0,42	4680	1130	486	264	164	66,0	34,0	19,6	12,2	5,50	2,70	1,34	0,66	0,31	0,11	0,02
1,0	0,24	4260	1030	443	240	149	60,0	31,0	17,8	11,1	5,00	2,40	1,20	0,61	0,29	0,11	0,02
1,4	0,10	3930	950	408	221	137	55,6	28,4	16,4	10,3	4,60	2,25	1,15	0,58	0,28	0,11	0,03
2,0	0,02	3770	910	391	212	134	53,0	27,4	15,8	9,90	4,40	2,20	1,13	0,58	0,28	0,12	0,04
3,0	0	3765	913	392	214	132	53,5	27,5	15,9	10,0	4,50	2,24	1,17	0,61	0,31	0,15	0,06
4,0	0	3775	930	400	215	132	53,8	27,7	16,2	10,0	4,60	2,25	1,20	0,64	0,35	0,16	0,08
5,0	0	3850	936	400	220	133	55,5	28,5	16,5	10,5	4,75	2,40	1,28	0,69	0,37	0,20	0,10
6,0	0	3870	940	400	222	133	55,8	28,5	16,6	10,5	4,80	2,42	1,32	0,70	0,40	0,21	0,12
7,0	0	4000	950	405	230	135	55,9	29,0	17,0	10,9	5,00	2,50	1,38	0,74	0,43	0,23	0,14
8,0	0	4000	965	410	236	137	56,0	30,0	17,2	11,2	5,10	2,58	1,45	0,78	0,45	0,25	0,16
9,0	0	4080	985	420	240	140	57,0	30,0	17,4	11,4	5,30	2,62	1,50	0,80	0,50	0,28	0,18
10	0	4110	1000	430	245	146	59,7	31,0	18,2	11,5	5,40	2,80	1,57	0,89	0,53	0,32	0,20

To recalculate flow resistance in the throttle ξ_{thr} from ξ_{nom}, use the formula:

$$\xi_{thr} = \xi_{nom} \frac{A_0^2}{A_1^2} \quad (316)$$



Circular cross section

Rectangular cross section

Figure 159: Single Disk Gate Valves (Switch Orifices)

Table 9 contains values of the flow resistance coefficient ξ_{nom} related to the nominal cross-sectional area of orifice. For circular cross-section, ξ_{nom} values have to be selected according to the ratio between the open height and orifice diameter h/D_0 as shown in Figure 159. For rectangular cross-section, ξ_{nom} values have to be selected according to the ratio h/a_0 .

Table 9: Single Disk Gate Valves

ξ_{nom}	$h/D_0(h/a_0)$									
	0,10	0,15	0,20	0,30	0,40	0,5	0,6	0,7	0,8	0,9
	A_h/A_0 (for circular cross section)									
	-	-	0,25	0,38	0,50	0,61	0,71	0,81	0,90	0,96
Circular Cross Section	(D_0 = 25mm)									
Curve 1	-	-	35,0	10,0	4,60	2,06	0,98	0,44	0,17	0,06
Rectangular Cross Section										
0,5 (25 x 50). Curve 2	193	-	44,5	17,8	8,12	4,02	2,08	0,95	0,39	0,09
0,5 (150 x 300 mm).Curve 3	105	51,5	30,6	13,5	6,85	3,34	1,73	0,83	0,32	0,09
1,0 (150 x 150 mm).Curve 4	155	72,0	42,3	18,5	8,78	4,54	2,43	1,23	0,55	0,17
1,0 (225 x 150 mm).Curve 5	330	122	58,2	19,6	9,10	4,68	2,66	1,23	0,47	0,11
2,0 (300 x 150 mm).Curve 6	203	86,5	48,7	17,9	8,78	4,47	2,25	1,12	0,51	0,13

To recalculate ξ_{thr} from ξ_{nom}, use the formula:

$$\xi_{thr} = \xi_{nom} \frac{A_h^2}{A_I^2} \tag{317}$$

Table 10 contains values of flow resistance coefficients at input/output connections ξ_{in} and ξ_{out} for sudden contractions. Resistance coefficient values have to be selected according to the ratio between the cross-sectional areas of input/output connection and orifice tube A_0/A_1 and Reynolds number.

For contraction at input end $A_0/A_1 = A_{nom}/A_{in}$.

For contraction at output end $A_0/A_1 = A_{nom}/A_{out}$.

Table 10: Sudden Contraction, Expansion

ξ_{inp}, ξ_{out}	Re													
A_0/A_1	10	20	30	40	50	10^2	$2x10^2$	$5x10^2$	10^3	$2x10^3$	$4x10^3$	$5x10^3$	10^4	$>10^4$
0,1	5,00	3,20	2,40	2,00	1,80	1,30	1,04	0,82	0,64	0,50	0,80	0,75	0,50	0,45
0,2	5,00	3,10	2,30	1,84	1,62	1,20	0,95	0,70	0,50	0,40	0,60	0,60	0,40	0,40
0,3	5,00	2,95	2,15	1,70	1,50	1,10	0,85	0,60	0,44	0,30	0,55	0,55	0,35	0,35
0,4	5,00	2,80	2,00	1,60	1,40	1,00	0,78	0,50	0,35	0,25	0,45	0,50	0,30	0,30
0,5	5,00	2,70	1,80	1,46	1,30	0,90	0,65	0,42	0,30	0,20	0,40	0,42	0,25	0,25
0,6	5,00	2,60	1,70	1,35	1,20	0,80	0,56	0,35	0,24	0,15	0,35	0,35	0,20	0,20

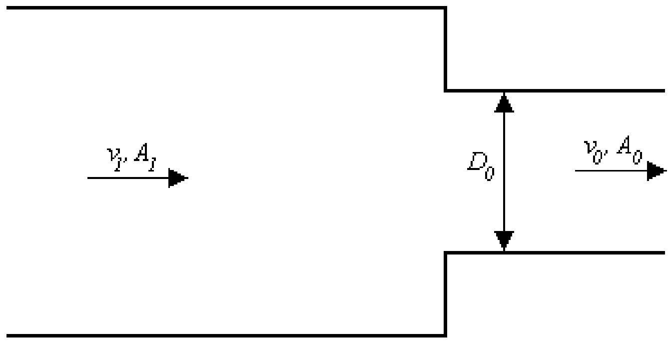


Figure 160: Sudden Contraction

Reynolds number is given by the formula:

$$Re = \frac{v_0 D_h}{\upsilon} \tag{318}$$

where v_0 is the flow velocity and ν is the kinematic viscosity.

Flow coefficients ξ_{in} and ξ_{out} at sudden contraction (refer to Figure 160) should be chosen according to the following rule:

- for 10 < Re > 10^4 : from Table 10
- for 1 < Re < 6-7 : from the formula $\xi = \frac{30}{\text{Re}}$.

Table 8, Table 9, and Table 10 contain flow resistance coefficients only for a short range of orifice parameters. The complete tables can be found in the reference[30].

[30] Idelchik, I.E. *Handbook of Hydraulic Resistance*. Springer-Verlag, 2nd Edition, 1986.

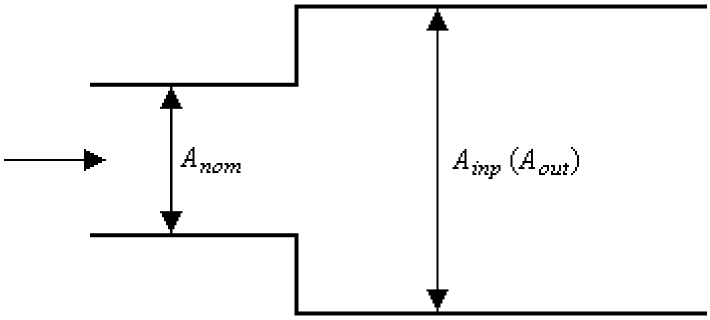


Figure 161: Sudden Expansion

For the calculation of resistance coefficients ξ_{in} and ξ_{out} at sudden expansion (refer to Figure 161) Carnot formula can be used if no better estimates are available:

$$\xi_{in} = \frac{1}{2}\left(1 - \frac{A_{nom}}{A_{in}}\right) \; ; \; \xi_{out} = \left(1 - \frac{A_{nom}}{A_{out}}\right)^2 \tag{319}$$

19.1.6. Isothermal Flow Equations

Flow through orifice occurs at high Reynolds numbers and is often referred as “turbulent” flow. However, the term “turbulent flow” does not have quite the same meaning here as in pipe flow. The flow from input to output end of orifice is a streamline or potential flow. Experience justifies the use of Bernoulli equations in this region.

In BOOST Hydsim model of **Standard Orifice** (refer to Figure 157), Bernoulli equation is applied up to three times: at input Expansion/Contraction (if defined), at the Throttle and at output Expansion/Contraction (if defined). Flow in positive or forward direction (from input to output) and negative or backward direction (from output to input) is considered separately as shown below.

1) Flow through input end

Positive (forward) flow ($p_{in} > p_1$)

$$\frac{p_{in}}{\rho} + \frac{v_{in}^2}{2} = \frac{p_1}{\rho} + \frac{v_1^2}{2} + \xi_{in+}\frac{v_1^2}{2},$$

$$p_{in} - p_1 = \frac{\rho}{2}\dot{Q}_+^2 \underbrace{\left[\frac{1+\xi_{in+}}{A_{nom}^2} - \frac{1}{A_{in}^2}\right]}_{\chi_{in}} \tag{320}$$

Negative (backward) flow ($p_{in} < p_1$)

$$\frac{p_{in}}{\rho} + \frac{v_{in}^2}{2} = \frac{p_1}{\rho} + \frac{v_1^2}{2} - \xi_{in-}\frac{v_1^2}{2},$$

$$p_1 - p_{in} = \frac{\rho}{2}\dot{Q}_-^2 \underbrace{\left[\frac{1-\xi_{in-}}{A_{nom}^2} - \frac{1}{A_{in}^2}\right]}_{\chi_{in}} \tag{321}$$

2) Flow through throttle

$$\frac{p_1}{\rho} + \frac{v_1^2}{2} = \frac{p_2}{\rho} + \frac{v_2^2}{2} \pm \xi_{thr}\frac{v_{thr}^2}{2}, \tag{322}$$

Positive (forward) flow ($p_1 > p_2$)

$$p_1 - p_2 = \frac{\rho}{2}\dot{Q}_+^2 \underbrace{\left[\frac{\xi_{thr}}{A_{thr}^2}\right]}_{\chi_{thr}}. \tag{323}$$

Negative (backward) flow ($p_1 < p_2$)

$$p_1 - p_2 = -\frac{\rho}{2}\dot{Q}_-^2 \underbrace{\left[\frac{\xi_{thr}}{A_{thr}^2}\right]}_{\chi_{thr}}. \tag{324}$$

3) Flow through output end

Positive (forward) flow ($p_2 > p_{out}$)

$$\frac{p_2}{\rho} + \frac{v_2^2}{2} = \frac{p_{out}}{\rho} + \frac{v_{out}^2}{2} + \xi_{out+}\frac{v_2^2}{2},$$

$$p_2 - p_{out} = -\frac{\rho}{2}\dot{Q}_+^2 \underbrace{\left[\frac{1-\xi_{out+}}{A_{nom}^2} - \frac{1}{A_{out}^2}\right]}_{\chi_{out}}. \tag{325}$$

Negative (backward) flow ($p_2 < p_{out}$)

$$\frac{p_2}{\rho} + \frac{v_2^2}{2} = \frac{p_{out}}{\rho} + \frac{v_{out}^2}{2} - \xi_{out-}\frac{v_2^2}{2},$$

$$p_{out} - p_2 = \frac{\rho}{2}\dot{Q}_-^2 \underbrace{\left[\frac{1+\xi_{out-}}{A_{nom}^2} - \frac{1}{A_{out}^2}\right]}_{\chi_{out}}. \tag{326}$$

with:

$\dot{Q}_+$	forward flow rate through orifice (from input to output)
$\dot{Q}_-$	backward flow rate through orifice (from output to input)
p_{in}	input pressure
p_1	pressure before throttle (upstream)

p_2	pressure after throttle (downstream)
p_{out}	output pressure
v_{in}	velocity of fluid on input end
v_1	velocity of fluid before throttle
v_{thr}	velocity of fluid in "vena contracta" (at throttle exit)
v_2	velocity of fluid after throttle
$\mathrm{v_{out}}$	velocity of fluid on output end
ξ_{in+}	forward flow resistance coefficient at input end
ξ_{in-}	backward flow resistance coefficient at input end
ξ_{thr}	flow resistance coefficient in the throttle
ξ_{out+}	forward flow resistance coefficient at output end
ξ_{out-}	backward flow resistance coefficient at output end

For different types of orifices, the flow resistance coefficients could be found in the tables of Section 19.1.5 and in literature [e.g. Idelchik].

Adding Equations 320, 322 and 324 and rearranging terms, we obtain the positive flow rate through **Standard Orifice**:

$$\dot{Q}_+ = \sqrt{\frac{1}{\chi_{in} + \chi_{thr} - \chi_{out}}}\sqrt{\frac{2}{\rho}(p_{in} - p_{out})}\ , \tag{327}$$

Analogously, adding Equations 321, 323 and 325 and rearranging terms, we obtain the negative flow rate through Standard Orifice:

$$\dot{Q}_- = \sqrt{\frac{1}{\chi_{out} + \chi_{thr} - \chi_{in}}}\sqrt{\frac{2}{\rho}(p_{out} - p_{in})}\ , \tag{328}$$

Clearly, if there are no Contractions/Expansions at Orifice ends, Equations 327 and 328 and reduce to:

$$\dot{Q}_\pm = sign(p_{in} - p_{out})\sqrt{\frac{1}{\chi_{thr}}\frac{2}{\rho}\left|p_{in} - p_{out}\right|}\ . \tag{329}$$

19.1.7. Thermal Flow Equations

The implementation of thermodynamic calculation involves the following specific assumptions:

1. Inflow pressure and temperature as well as outflow pressure are known parameters. These parameters are taken from the connected elements.

2. For liquids, specific heat at constant pressure and specific heat at constant volume are almost the same, so it is assumed they are the same.
3. Fluid is a compressible liquid whose density (specific volume) is temperature and pressure dependent.
4. Fluid flow through the Orifice is considered as ideal from the inlet side to the throttle (abrupt contraction) and energy losses are considered only in the outlet part of the Orifice (from the throttle to the outlet side (abrupt enlargement)).
5. Orifice is split into four parts (refer to Figure 162):
   - from inlet side to point before throttle (contraction from inp to 1)
   - from the point before throttle to the throttle (contraction from 1 to 2)
   - from the throttle to the point after throttle (expansion from 2 to 3)
   - from the point after throttle to outlet side (expansion from 3 to out)
6. There is no heat exchange between Orifice and Ambient (Adiabatic flow).
7. Orifice length is neglected.

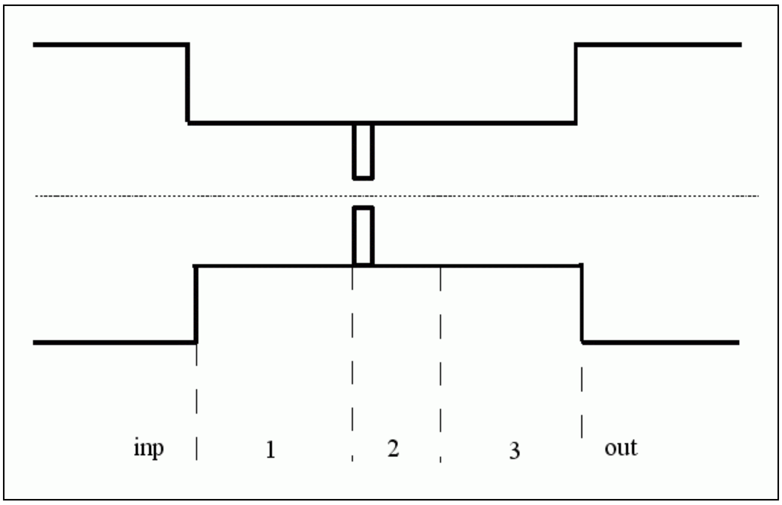


Figure 162: Orifice Split into 4 Sections

According to the assumption 5, all equations through the Orifice may be written within four sections:

1. Section from Inlet side to point 1:
   – equations used are **Conservation of mass** and **Bernoulli equation** (energy equation for the ideal flow)
2. Section from point 1 to point 2:
   – equations used are **Conservation of mass** and **Bernoulli equation** (energy equation for the ideal flow)
3. Section from point 2 to point 3:
   – equations used are **Conservation of mass** , **Momentum equation** and **Energy equation**
4. Section from point 3 to outlet side:
   – equations used are **Conservation of mass** , **Momentum equation** and **Energy equation**

19.1.7.1. Fluid flow from INP to 1:

Fluid flow from section INP to section 1 can be considered as ideal, i.e. Bernoulli equation can be used.

$$T_1 = T_{inp}$$

Conservation of mass:

$$\rho_{inp} u_{inp} A_{inp} = \rho_1 u_1 A_1$$

$$u_1 = \frac{\rho_{inp}}{\rho_1} \frac{A_{inp}}{A_1} u_{inp} = C_1 \cdot u_{inp} , \tag{330}$$

where C_1 is constant:

$$C_1 = \frac{\rho_{inp}}{\rho_1} \frac{A_{inp}}{A_1}$$

A_{inp}	-	cross sectional area on inlet side (user-defined parameter)
A_1	-	effective tube cross sectional area (before throttle)

$$A_1 = (c_c)_{inp} A_{tube}$$

$(c_c)_{inp}$	-	contraction coefficient of input side
A_{tub}	-	tube cross sectional area (user-defined parameter)

$$\mu_{inp} \approx (c_c)_{inp}$$

$$\mu_{inp} = \frac{1}{\sqrt{\varsigma_{inp}}}$$

μ_{inp}	-	discharge coefficient
ζ_{inp}	-	flow resistance coefficient at input end (user-defined parameter)

Energy (Bernoulli) equation:

$$\frac{p_{inp}}{\rho_{inp}} + \frac{u_{inp}^2}{2} = \frac{p_1}{\rho_1} + \frac{u_1^2}{2} ,$$

where:

ρ_{inp}	-	fluid density at inlet side; $f(p_{inp}, T_{inp})$
u_{inp}	-	fluid flow velocity at inlet side

ρ_1 - fluid density at point 1 (p_1 from previous time step); $f(p_1, T_1)$

u_1 - fluid flow velocity at point 1 (u_1 can be calculated from eq. 330)

$$p_1 = \frac{\rho_1}{\rho_{inp}} p_{inp} + u_{inp}^2 \left(\frac{\rho_1}{2} - \frac{\rho_1 C_1^2}{2} \right),$$

$$\boxed{p_1 = C_{p1} p_{inp} + C_{u1} u_{inp}^2}, \tag{331}$$

where C_{p1} and C_{u1} are constants:

$$C_{p1} = \frac{\rho_1}{\rho_{inp}},$$

$$C_{u1} = \frac{\rho_1}{2} - \frac{\rho_1 C_1^2}{2} = \frac{\rho_1}{2}\left(1 - C_1^2\right) = \frac{\rho_1}{2}\left[1 - \left(\frac{\rho_{inp}}{\rho_1}\frac{A_{inp}}{A_1}\right)^2\right].$$

19.1.7.2. Fluid flow from 1 to 2:

Fluid flow from section 1 to section 2 can be considered ideal, i.e. Bernoulli equation can be used.

$$T_2 = T_1 = T_{inp}$$

Conservation of mass:

$$\rho_1 u_1 A_1 = \rho_2 u_2 A_2,$$

$$u_2 = \frac{\rho_1}{\rho_2}\frac{A_1}{A_2} u_1 = C_2 u_1,$$

$$u_2 = C_1 C_2 u_{inp} = C_{12} u_{inp}, \tag{332}$$

where C_{12} is constant:

$$C_{12} = \frac{\rho_1}{\rho_2}\frac{A_1}{A_2}\frac{\rho_{inp}}{\rho_1}\frac{A_{inp}}{A_1} = \frac{\rho_{inp}}{\rho_2}\frac{A_{inp}}{A_2},$$

A_{inp} - cross sectional area of input expansion (user-defined parameter)

A_2 - vena-contracta cross sectional area

$$A_2 = c_c A_{throttle}$$

c_c	-	contraction coefficient of throttle
$A_{throttle}$	-	throttle cross sectional area (user-defined parameter)

$\mu \approx c_c$

$\mu = \frac{1}{\sqrt{\varsigma}}$

μ	-	discharge coefficient
ζ	-	flow resistance coefficient at throttle (user-defined parameter)

Energy (Bernoulli) equation:

$$\frac{p_1}{\rho_1} + \frac{u_1^2}{2} = \frac{p_2}{\rho_2} + \frac{u_2^2}{2},$$

ρ_1	-	fluid density at point 1 (p1 from previous time step); f(p1, T1)
u_1	-	fluid flow velocity at point 1 (u1 can be calculated from eq. 330)
ρ_2	-	fluid density at point 2 (p2 from previous time step); f(p2, T2)
u_2	-	fluid flow velocity at point 2 (u2 can be calculated from eq. 332)
p_1	-	pressure at point 1 (from eq. 331)

$$\frac{p_2}{\rho_2} = \frac{p_1}{\rho_1} + \frac{u_1^2}{2} - \frac{u_2^2}{2},$$

$$\boxed{p_2 = C_{p2} p_{inp} + C_{u2} u_{inp}^2}, \qquad (333)$$

where C_{p2} and C_{u2} are constants:

$$C_{p2} = \frac{\rho_2}{\rho_1} C_{p1} = \frac{\rho_2}{\rho_1} \frac{\rho_1}{\rho_{inp}} = \frac{\rho_2}{\rho_{inp}},$$

$$C_{u2} = \rho_2 \frac{1 - C_{12}^2}{2} = \rho_2 \frac{1 - \left(\frac{\rho_{inp}}{\rho_2} \frac{A_{inp}}{A_2} \right)^2}{2}.$$

19.1.7.3. Fluid flow from 2 to 3:

Fluid flow from point 2 to point 3 can be considered as flow at abrupt expansion, so energy losses must be considered.

Conservation of mass:

$$\rho_2 u_2 A_2 = \rho_3 u_3 A_3 ,$$

$$u_3 = \frac{\rho_2}{\rho_3}\frac{A_2}{A_3}u_2 = C_3 u_2 ,$$

$$u_3 = C_{12} C_3 u_{inp} = C_{13} u_{inp} , \quad (334)$$

where C_{13} is constant:

$$C_{13} = \frac{\rho_2}{\rho_3}\frac{A_2}{A_3}\frac{\rho_{inp}}{\rho_2}\frac{A_{inp}}{A_2} = \frac{\rho_{inp}}{\rho_3}\frac{A_{inp}}{A_3} ,$$

A_{inp} - cross sectional area of input expansion (user-defined parameter)

A_3 - effective tube cross sectional area after throttle (user-defined parameter)

Momentum equation:

$$(p_2 - p_3)A_3 = \rho_3 u_3 A_3 (u_3 - u_2),$$

u_2 - fluid flow velocity at point 2 (u_2 can be calculated from eq. 332)

u_3 - fluid flow velocity at point 3 (u_3 can be calculated from eq. 334)

$$p_3 = p_2 - \rho C_{13} u_{inp} \left(C_{13} u_{inp} - C_{12} u_{inp}\right),$$

p_2 - pressure at point 2 (from eq. 333)

ρ_3 - fluid density at point 3 (p_3 and T_3 from previous time step); $f(p_3, T_3)$

u_{inp} - fluid flow velocity at input side

$$\boxed{p_3 = C_{p2} p_{inp} + C_{u3} u_{inp}^2}, \quad (335)$$

where C_{p2} (refer to previous section) and C_{u3} are constants:

$$C_{u3} = \rho_2 \frac{1 - \left(\frac{\rho_{inp}}{\rho_2}\frac{A_{inp}}{A_2}\right)^2}{2} - \rho_3 \frac{\rho_{inp}}{\rho_3}\frac{A_{inp}}{A_3}\left(\frac{\rho_{inp}}{\rho_3}\frac{A_{inp}}{A_3} - \frac{\rho_{inp}}{\rho_2}\frac{A_{inp}}{A_2}\right).$$

Energy equation:

$$c_2 T_2 + \frac{p_2}{\rho_2} + \frac{u_2^2}{2} = c_3 T_3 + \frac{p_3}{\rho_3} + \frac{u_3^2}{2} ,$$

c_2 - $f(T_2) - T_2$ from previous time step
c_3 - $f(T_3) - T_3$ from previous time step

$$c_3 T_3 = c_2 T_2 + \frac{p_2}{\rho_2} - \frac{p_3}{\rho_3} + \frac{u_2^2}{2} - \frac{u_3^2}{2},$$

$$\boxed{T_3 = C_{tt3} T_{inp} + C_{tp3} p_{inp} + C_{tu3} u_{inp}^2}, \qquad (336)$$

where C_{tt3}, C_{tp3} and C_{tu3} are constants:

$$C_{tt3} = \frac{c_2}{c_3},$$

$$C_{tp3} = \frac{1}{\rho_{inp} c_3}\left(\frac{1}{\rho_2} - \frac{1}{\rho_3}\right),$$

$$C_{tu3} = \frac{1}{c_3}\left(\frac{1}{2} - \frac{\rho_2\left(1 - \left(\frac{\rho_{inp}}{\rho_2}\frac{A_{inp}}{A_2}\right)^2\right)}{\rho_3} + \frac{\left(\frac{\rho_{inp}}{\rho_3}\frac{A_{inp}}{A_3}\right)^2}{2} - \frac{\rho_{inp}}{\rho_2}\frac{A_{inp}}{A_2} \cdot \frac{\rho_{inp}}{\rho_3}\frac{A_{inp}}{A_3}\right).$$

19.1.7.4. Fluid flow from 3 to OUT:

Fluid flow from point 3 to output side can be considered as flow at abrupt expansion, so energy losses must be considered.

<u>Mass equation:</u>

$$\rho_3 u_3 A_3 = \rho_{out} u_{out} A_{out},$$

$$u_{out} = \frac{\rho_3}{\rho_{out}}\frac{A_3}{A_{out}} u_3 = C_4 u_3,$$

$$u_{out} = C_{13} C_4 u_{inp} = C_{14} u_{inp}, \qquad (337)$$

where C_{14} is constant:

$$C_{14} = \frac{\rho_3}{\rho_{out}}\frac{A_3}{A_{out}}\frac{\rho_{inp}}{\rho_3}\frac{A_{inp}}{A_3} = \frac{\rho_{inp}}{\rho_{out}}\frac{A_{inp}}{A_{out}}$$

A_{inp} - cross sectional area of input expansion (user-defined parameter)

A_{out} - cross sectional area of output expansion (user-defined parameter)

$$A_{out} = (c_c)_{out} A_{out'}$$

$(c_c)_{out}$ - contraction coefficient of output side

$\mu_{out} \approx (c_c)_{out}$

$\mu_{out} = \dfrac{1}{\sqrt{\varsigma_{out}}}$

μ_{out} - discharge coefficient

ζ_{out} - flow resistance coefficient at output side (user-defined parameter)

Momentum equation:

$$(p_3 - p_{out})A_{out} = \rho_{out} u_{out} A_{out} (u_{out} - u_3),$$

p_3 - pressure at point 2 (from eq. 335)

u_{inp} - fluid flow velocity at input side

ρ_{out} - fluid density at output side (T_{out} from previous time step); $f(p_{out}, T_{out})$

$$C_{p2} p_{inp} + C_{u3} u_{inp}^2 - p_{out} = \rho_{out} C_{14} u_{inp}^2 (C_{14} - C_{13}),$$

$$\boxed{u_{inp} = \sqrt{\frac{C_{p2} p_{inp} - p_{out}}{[\rho_{out} C_{14}(C_{14} - C_{13}) - C_{u3}]}}}, \tag{338}$$

Equation 338 into:

eq. 337	=>	u_{out}
eq. 336	=>	T_3
eq. 335	=>	p_3
eq. 334	=>	u_3
eq. 333	=>	p_2
eq. 331	=>	p_1

Energy equation:

$$c_3 T_3 + \frac{p_3}{\rho_3} + \frac{u_3^2}{2} = c_{out} T_{out} + \frac{p_{out}}{\rho_{out}} + \frac{u_{out}^2}{2},$$

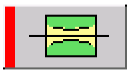
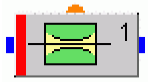
c_{out} - $f(T_{out})$ – T_{out} from previous time step

$$c_{out} T_{out} = c_3 T_3 + \frac{p_3}{\rho_3} - \frac{p_{out}}{\rho_{out}} + \frac{u_3^2}{2} - \frac{u_{out}^2}{2},$$

$$\boxed{T_{out} = \frac{c_3}{c_{out}} T_3 + \frac{p_3}{c_{out}\rho_3} - \frac{p_{out}}{c_{out}\rho_{out}} + \frac{u_3^2}{2c_{out}} - \frac{u_{out}^2}{2c_{out}}} \qquad (339)$$

T_{out} - fluid temperature at output side

19.2. Cavitating Orifice

Element Name:	**Cavitating Orifice**	
Element Icon:		
Definition:	This element serves to define the properties of Cavitating Orifice (laminar/turbulent/cavitating flow model).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Note: **Cavitating Orifice** is basically analogous to the **Standard Orifice** except for the cavitating flow condition introduced there.

Note: Turbulent and cavitating flow resistance in **Cavitating Orifice** can be defined by entering the flow coefficient values at turbulent (hydraulic) and cavitating flow or the corresponding flow rates and pressure difference across the orifice. In this case both flow resistance coefficients are calculated directly in GUI window.

Schematic of the **Cavitating Orifice** with expansions/contractions at connections is shown in Figure 163.

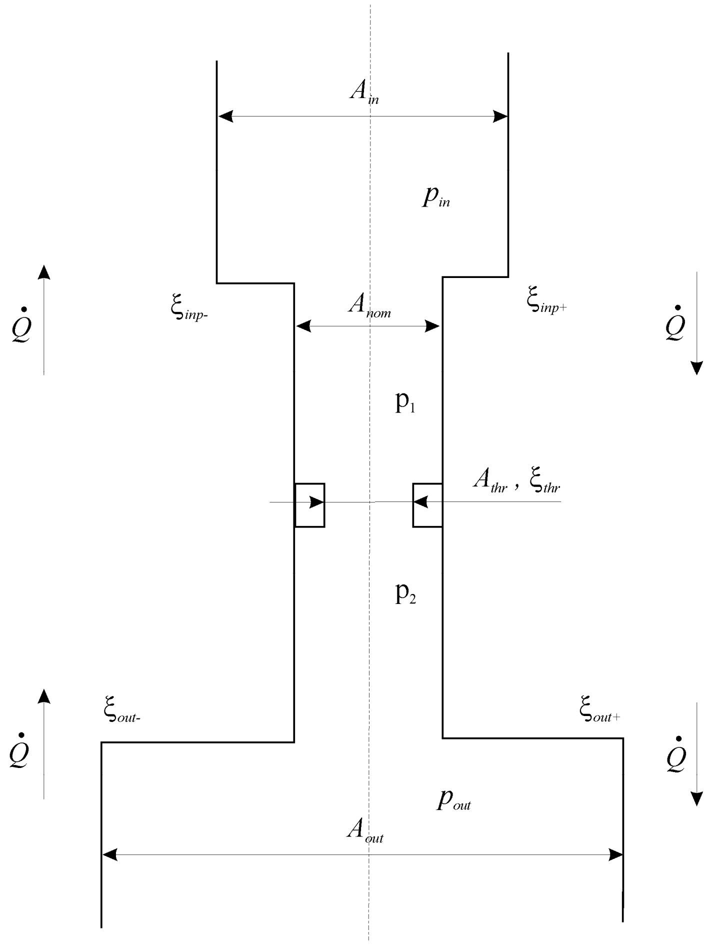


Figure 163: Cavitating Orifice Geometry

19.2.1. Input Parameters

Description of input data for the **Cavitating Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
CROSS-SECTION AREA	
Tube cross-section area	Specify the cross-sectional area of orifice tube A_{nom} (refer to Figure 163).
Throttle cross-section area	Specify cross-sectional area of throttle A_{thr} (refer to Figure 163).
CROSS-SECTION DIAMETER	
Tube cross-section diameter	Specify the cross-sectional diameter of orifice tube d_{nom}.
Throttle cross-section diameter	Specify cross-sectional diameter of throttle d_{thr}.

FLOW COEFFICIENT IN THROTTLE (TURBULENT/CAVITATING)	
Critical flow number (laminar->turbulent)	Specify critical flow number (λ) for transition between laminar and turbulent flow
Critical cavitation number	Specify critical cavitation number ( ) for transition between turbulent and cavitating flow. If it is active, then turbulent flow resistance/discharge coef. Of flow rate input s disabled
Flow Resistance coef.	For this option the resistance coefficients at turbulent and cavitating flow have to be specified.
Turbulent flow resistance (>1)	Specify flow resistance coefficient ξ_{thr} in throttle (narrowest cross-sectional area) at turbulent (hydraulic) flow. Active only if check box Critical cavitation number is disabled For estimation of ξ_{thr} , refer to Section 19.1.5.
Cavitating flow resistance (>1)	Specify flow resistance coefficient ξ_{thr} in throttle (narrowest cross-sectional area) at cavitating flow.
Flow Rate	For this option flow rates through orifice has to be specified. Hydraulic/cavitating flow resistance coefficients are then calculated from Bernoulli equation: $$\mu_{hyd/cav} = \frac{Q_{hyd/cav\_m}}{A_{thr}} \sqrt{\frac{\rho}{2\Delta p_{hyd/cav\_m}}}$$ - if throttle cross-section area is activated $$\mu_{hyd/cav} = \frac{4Q_{hyd/cav\_m}}{\pi d_{thr}^2} \sqrt{\frac{\rho}{2\Delta p_{hyd/cav\_m}}}$$ - if throttle cross-section diameter is activated
Calculate coef.	Press to calculate the flow resistance coefficients and display them at Z_{hyd} and Z_{cav} positions: $$Z_{hyd} = \frac{1}{\mu_{hyd}^2}, \quad Z_{cav} = \frac{1}{\mu_{cav}^2}$$
Turbulent flow rate	Specify measured (or obtained otherwise) flow rate through throttle at turbulent (hydraulic) flow Q_{hyd_m}. Active only if check box Critical cavitation number is disabled
Cavitating flow rate	Specify measured (or obtained otherwise) flow rate through throttle at cavitating flow Q_{cav_m}
Pressure input-output	Specify pressure difference Δp_{hyd_m} at which volumetric flow is measured (or obtained otherwise).
Pressure input-vapor	Specify pressure Δp_{cav_m} at which volumetric flow is measured (or obtained otherwise).
Discharge coef.	For this option the rdischarge coefficients at turbulent and cavitating flow have to be specified.

Turbulent flow discharge (<1)	Specify flow discharge coefficient μ_{thr} in throttle (narrowest cross-sectional area) at turbulent flow. Active only if check box Critical cavitation number is disabled
Cavitating flow discharge (<1)	Specify flow discharge coefficient μ_{thr} in throttle (narrowest cross-sectional area) at cavitating flow
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS	
Cross-sectional area at input end	Specify cross-sectional area at input end A_{in}. **Note:** If cross-sectional area at input end is zero (empty), it implies $A_{in}= A_{nom}$.
Cross-sectional area at output end	Specify cross-sectional area at output end A_{out}. **Note:** If cross-sectional area at output end is zero (empty), it implies $A_{out}= A_{nom}$.
Positive flow resistance coefficient at input end	Specify resistance coefficient ξ_{int+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on input end. Active only if the **Expansions/Contractions at Connections** box is checked.
Positive flow resistance coefficient at output end	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output end. Active only if the **Expansions/Contractions at Connections** box is checked.
Negative flow resistance coefficient at input end	Specify resistance coefficient ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on input end. Active only if the **Expansions/Contractions at Connections** box is checked.
Negative flow resistance coefficient at output end	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end. Active only if the **Expansions/Contractions at Connections** box is checked.

19.2.2. Initial Conditions

Initial conditions cannot be specified for **Cavitating Orifice** element.

19.2.3. Modifiable Parameter

Path: **Element| Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

tube cross-section area	SI Unit:	m^2
throttle cross-section area	SI Unit:	m^2
hydraulic flow resistance coefficient	SI Unit:	---
cavitating flow resistance coefficient	SI Unit:	---
cross-section area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-section area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---
tube cross-section diameter	SI Unit:	m
throttle cross-section diameter	SI Unit:	m

19.2.4. Output Parameters

Path: **Element| Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

static pressure before throttle (input end)	**Corresponds to pressure p1 refer to Figure 163.**
static pressure after throttle (output end)	**Corresponds to pressure p2 refer to Figure 163.**
volumetric flow rate at orifice input end	**To obtain it, fluid flow velocity is calculated by Equation 338 in Section 19.1.6 and multiplied with cross-sectional area at input end Ain.**
volumetric flow rate at orifice input end	**Output values will be 0 if the Time domain button is active in Calculation Control dialog box.**
cumulative flow rate at orifice input end	**Cumulative flow rate per calculation range.**
cavitation number in throttle	**This is defined by the relationship** $\frac{p_{in} - p_{out}}{p_{out}}$

cavitation factor in throttle	**This is defined by the relationship** $\frac{p_{in} - p_{out}}{p_{in}}$
volumetric flow rate at orifice output end	**To obtain it, fluid flow velocity is calculated by Equation 337 in Section 19.1.6 and multiplied with cross-sectional area at output end Aout.**
volumetric flow rate at orifice output end	**Output values will be 0 if the Time domain button is active in Calculation Control dialog box.**
cumulative flow rate at orifice output end	**Cumulative flow rate per calculation range.**
flow coefficient in throttle	**Throttle flow coefficient for laminar/turbulent/cavitating flow**
mean flow velocity in throttle	**Average flow velocity of fluid in throttle (narrowest cross-sectional area)**
throttle flow number	**Flow number value in the throttle**
Reynolds number	**Reynolds number value in the throttle**

19.2.5. Flow Equations

The BOOST Hydsim model of **Cavitating Orifice** is analogous to the model of **Standard Orifice** (refer to Chapter 19.1) except that it includes a cavitation model in the throttle. Depending on the cavitation condition (refer to Section 19.2.6), Bernoulli equation in the throttle is used either with a flow resistance coefficient at hydraulic flow or resistance coefficient with cavitating flow.

For different types of orifices, the flow resistance coefficient ξ_{thr} could be found in the tables of Section 19.1.5 and in literature [Idelchik[1]]. Resistance coefficient at cavitation ξ_{cav} can be obtained by experiment as described in Section 19.2.6.

In injection systems, fuel vapor pressure is negligibly low compared to the liquid phase pressures, therefore it can be neglected in flow rate calculation ($p_{vap} \approx 0$).

19.2.6. Cavitation Phenomenon

When the decreasing pressure reaches the liquid vapor pressure at certain location, cavitation begins. Basically, cavitation is boiling of the liquid and formation and collapse (implosion) of vapor bubbles. Cavitation progress is influenced by flow conditions, geometry of solid boundaries, and fluid properties.

There are two different mechanisms that cause cavitation in the injection equipment. The cavitation phenomenon resulting from these mechanisms are referred to as dynamically-induced and geometry-induced cavitation. Dynamically-induced cavitation occurs only in transient flow and is usually caused by pressure wave activity or valve movement. Geometry-induced cavitation can occur in the steady-state as well as in transient flow.

A series of experiments was carried out in AVL to identify the phenomena of cavitation inside the nozzle holes. During each experiment p_{in} was held constant and $p_{\_out}$ was gradually decreased from $p_{out} = p_{in}$ to $p_{out} \approx 0$.

In this way, cavitation pocket was established at the inside edge of hole. This geometry-induced cavitation region spread along the hole as p_{out} was decreased. In the meantime, the fluid velocity was measured. It was rising as shown in Figure 165 (dashed line). The change of velocity followed the equation:

$$p_{in} - p_{out} = \frac{1}{2}\xi\rho v_0^2 \tag{340}$$

where the resistance coefficient ξ depends on the orifice geometry, fluid viscosity, and turbulence in downstream region.

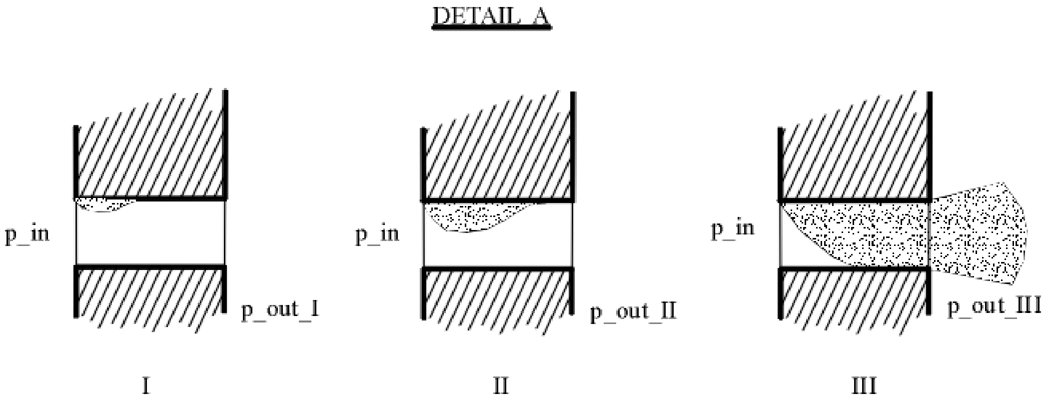


Figure 164: Cavitation Phenomenon in Orifice

At the moment when cavitation region approached the end of the hole (case III on Figure 164), any further decrease of the output pressure p_{out} did not lead to the rise of the fluid velocity in the spray hole. Hence Equation 299 is not valid in this region. The diagram in Figure 165 shows that only by increasing the input pressure p_{in} the fluid velocity inside spray holes could rise again.

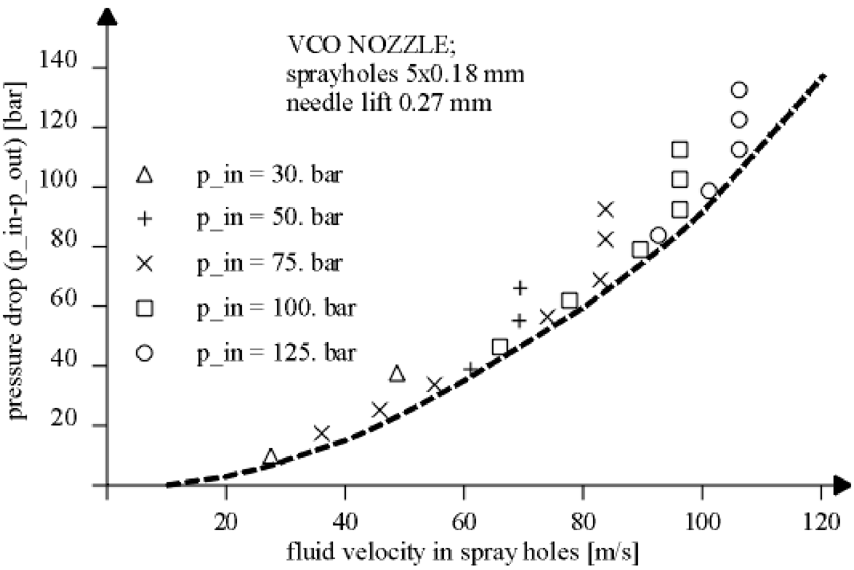


Figure 165: Relationship Between Pressure Drop and Fluid Velocity

Egartner[31] and Ofner and Egartner[32] showed that the cavitation phenomenon in the orifice could be described by the following equation:

[31] Egartner, Wolfgang. *Parameterschätzung in eindimensionalen Modellen für instationäre Strömung in Dieseleinspritzsystemen*. Dissertation, TU Graz, 1996.

$$p_{in} - p_{vap} = \frac{1}{2}\xi_{cav}\rho v_0^2 , \tag{341}$$

where p_{vap} is vapor pressure.

In the moment when full cavitation starts, both equations (hydraulic and cavitating) are valid. Hence at this point we have:

$$\frac{p_{in} - p_{out}}{p_{in} - p_{vap}} = \frac{\xi_{thr}}{\xi_{cav}} = C \tag{342}$$

where C is the cavitation factor given by:

$$C(p_{in}, p_{out}) = \frac{p_{in} - p_{out}}{p_{in}} \tag{343}$$

In Equation 343 it is assumed that $p_{vap} \approx 0$.

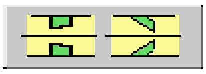
From Equation 342, the cavitation conditions can be established:

$$\frac{|p_{in} - p_{out}|}{p_{in}} \le \frac{\xi_{thr}}{\xi_{cav}} \Rightarrow \text{no cavitation} \tag{344}$$

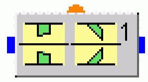
$$\frac{|p_{in} - p_{out}|}{p_{in}} \rangle \frac{\xi_{thr}}{\xi_{cav}} \Rightarrow \text{cavitation} \tag{345}$$

Note: In this section p_{in} corresponds to pressure before throttle and p_{out} corresponds to pressure after throttle.

19.3. Sharp-edged Orifice

Element Name:	**Sharp-edged Orifice**
Element Icon:	
Definition:	This element serves to define a Sharp-edged Orifice.

[32] Ofner, H. and Egartner, W. *Identification of Flow Phenomena in Fuel Injection Systems for Diesel Engines from Dynamic Measurements*. ImechE, Ch99/053, 1996.

Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Schematic of the Standard and Conical **Sharp-edged Orifice** with expansions/contractions at connections are shown in Figure 166 and Figure 167, respectively.

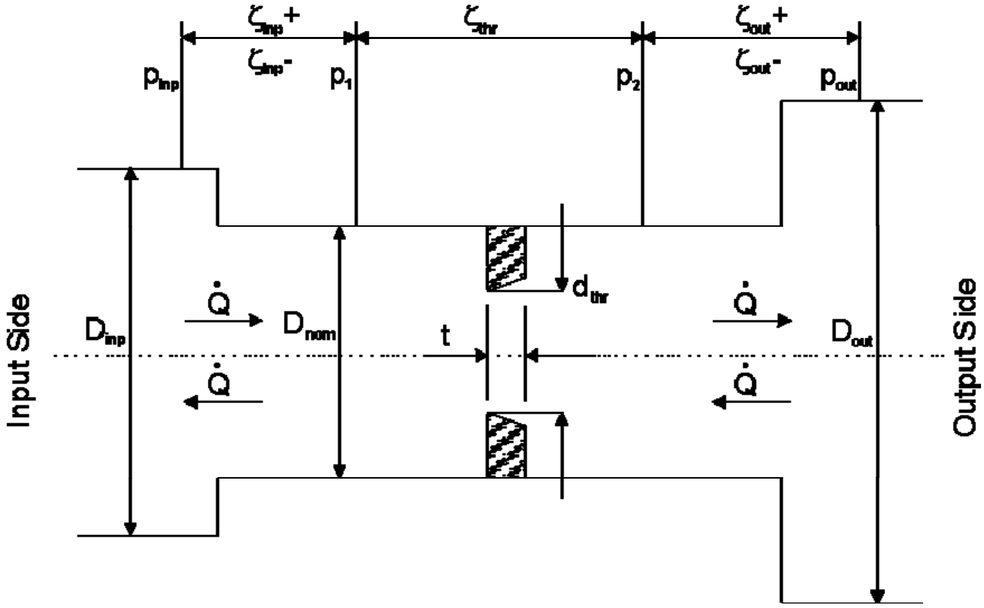


Figure 166: Standard Sharp-edged Orifice

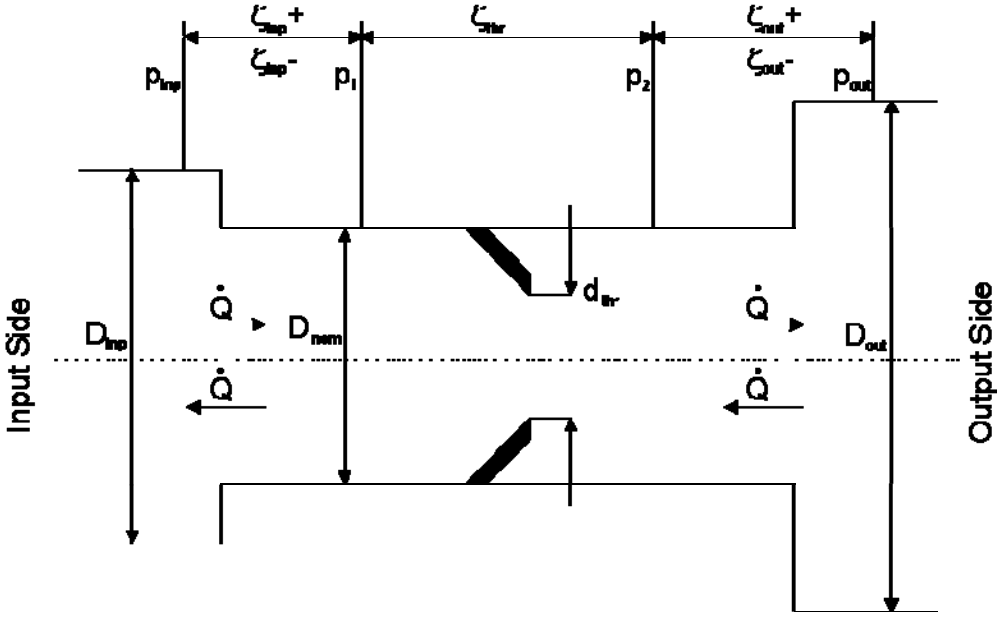


Figure 167: Conical Sharp-edged Orifice

19.3.1. Input Parameters

Description of input data for the **Sharp-edged Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Orifice type:	This button specifies the type of the sharp-edged orifice. There are two possible types: Standard sharp-edged orifice (refer to Figure 166) Conical sharp-edged orifice (refer to Figure 167) Flow resistance coefficient is estimated from internal database according to the orifice type.
Tube diameter	Specify tube diameter of the Orifice D_{nom} (refer to Figure 166 and Figure 167).
Throttle diameter	Specify diameter of the throttle d_{thr} (refer to Figure 166 and Figure 167).
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS	
Cross-sectional area at input end	Specify cross-sectional area at input end ($\frac{1}{4}D_{in}^2\pi$). **Note:** If cross-sectional area at input end is zero (empty), it implies $D_{in} = D_{nom}$.
Cross-sectional area at output end	Specify cross-sectional area at output end ($\frac{1}{4}D_{out}^2\pi$). **Note:** If cross-sectional area at output end is zero (empty), $D_{out} = D_{nom}$.
Positive flow resistance coefficient at input end	Specify resistance coefficient ξ_{int+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Positive flow resistance coefficient at output end	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output end.
Negative flow resistance coefficient at input end	Specify resistance coefficient ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Negative flow resistance coefficient at output end	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end.

19.3.2. Initial Conditions

Initial conditions cannot be specified for **Sharp-edged Orifice** element.

19.3.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

cross-section area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-section area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---

19.3.4. Output Parameters

Path: **Element**| **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

static pressure before throttle (input end)	Corresponds to pressure p_1: refer to Figure 166.
static pressure after throttle (output end)	Corresponds to pressure p_2 refer to Figure 166.
volumetric flow rate at orifice input end	Fluid flow velocity is calculated by Equation 338 in Section 19.1.6 and multiplied with cross-sectional area at input end A_{in}.
volumetric flow rate at orifice input end	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog.
cumulative flow rate at orifice input end	Cumulative flow rate per calculation range.
flow resistance coefficient in throttle	Internally calculated flow resistance coefficient.
volumetric flow rate at orifice output end	Fluid flow velocity is calculated by Equation 337 as given in Section 19.1.6 and multiplied with cross-sectional area at output end A_{out}.
volumetric flow rate at orifice output end	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog.
cumulative flow rate at orifice output end	Cumulative flow rate per calculation range.

19.3.5. Flow Equations

BOOST Hydsim model of **Sharp-edged Orifice** is analogous to the model of **Standard Orifice** (refer to Chapter **19.1**) except that the flow resistance coefficient ξ_{thr} is calculated as described in Section 19.3.6.

19.3.6. Estimation of Flow Resistance Coefficient

The flow resistance coefficient in the throttle ξ_{thr} is calculated from the flow resistance coefficient in the tube ξ_{nom} by the following formula:

$$\xi_{thr} = \xi_{nom} \frac{d_{thr}^4}{D_{nom}^4} \tag{346}$$

Flow resistance coefficient in the tube ξ_{nom} is calculated by multiplying the tabular flow resistance coefficient ξ_{tab} with the correction factor C_{Re} which depends on the flow regime:

$$\xi_{nom} = \xi_{tab} C_{\mathrm{Re}} \,. \tag{347}$$

Correction factor C_{Re} depends on the Reynolds number and is obtained from the diagram below.

Tabular flow resistance coefficient ξ_{tab} for the sharp-edged orifice is derived from the flow discharge coefficient μ according to the formula:

$$\xi_{tab} = \left[1 - \left(\frac{d_{thr}}{D_{nom}}\right)^2 \mu\right]^2 \frac{1}{\left(\frac{d_{thr}}{D_{nom}}\right)^4 \mu^2} \,. \tag{348}$$

Flow discharge coefficient μ for the sharp-edged orifice is obtained from the diagrams shown in Figure 168.[33]

[33] Miller D.S. *Internal Flow Systems*. 2nd Edition. BHRA (Information Services), 1990.

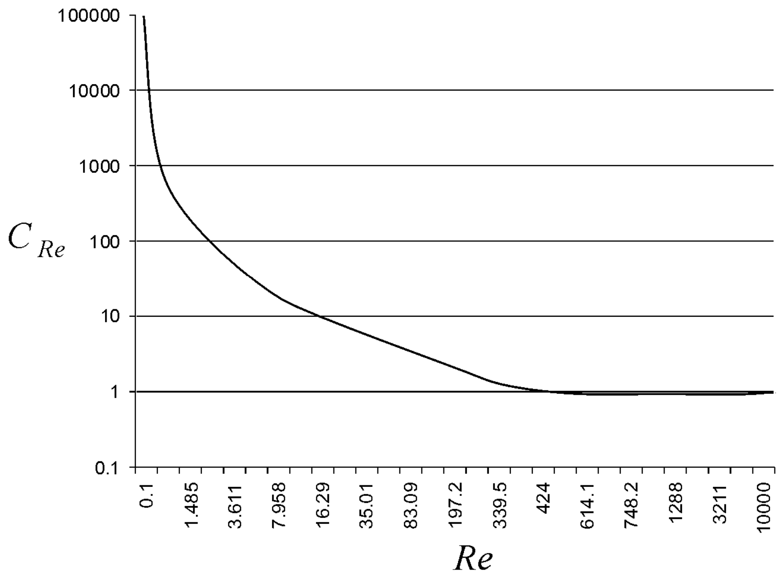


Figure 168: Correction Factor C_{Re} as Function of Reynolds Number Re

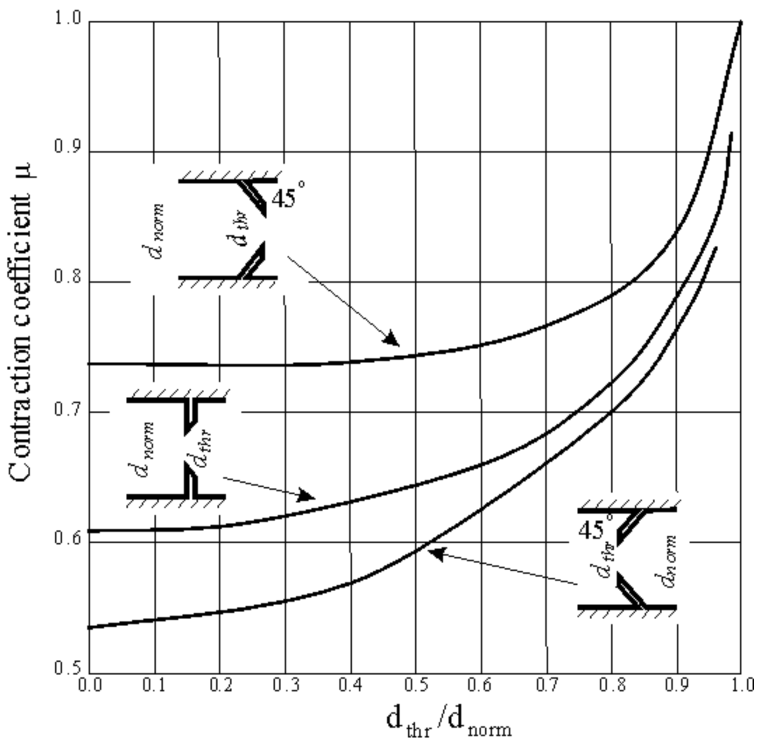
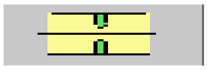
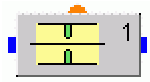


Figure 169: Flow Discharge Coefficient of Sharp-edged Orifice

19.4. Round-edged Orifice

Element Name:	**Round-edged Orifice**	
Element Icon:		
Definition:	This element serves to define a Round-edged Orifice.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Schematic of the **Round-edged Orifice** with expansions/contractions at connections is shown in Figure 170.

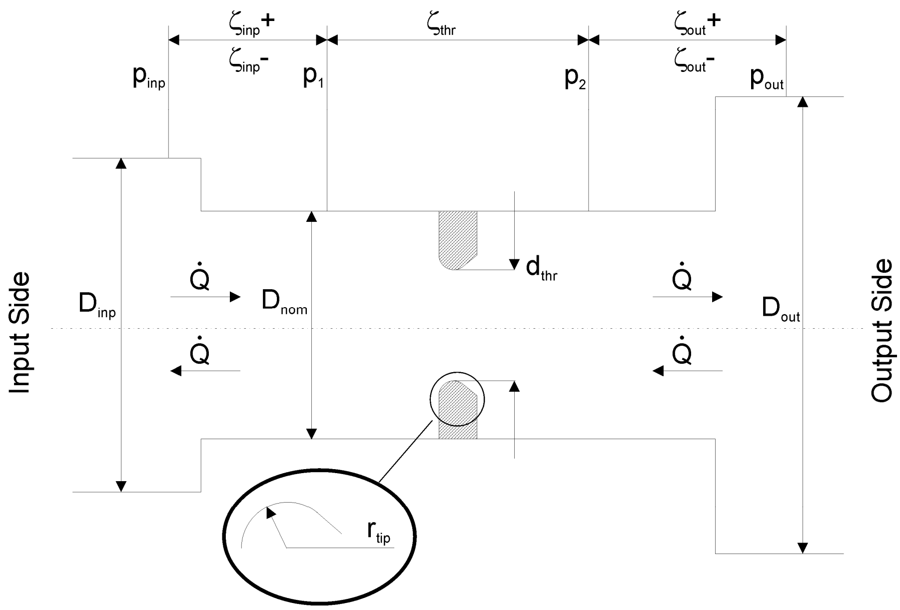


Figure 170: Round-edged Orifice

19.4.1. Input Parameters

Description of input data for the **Round-edged Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Tube diameter	Specify nominal diameter of Orifice tube D_{nom} (refer to Figure 170).
Throttle diameter	Specify diameter of throttle D_{thr} (refer to Figure 170).
Radius of throttle tip	Specify radius of the throttle tip r_{tip} (refer to Figure 170).
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS	
Cross-sectional area at input end	Specify cross-sectional area at input end ($\frac{1}{4} D_{in}^2 \pi$).
Cross-sectional area at output end	Specify cross-sectional area at output end ($\frac{1}{4} D_{out}^2 \pi$).
Positive flow resistance coefficient at input end	Specify resistance coefficient ξ_{int+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Positive flow resistance coefficient at output end	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output end.
Negative flow resistance coefficient at input end	Specify resistance coefficient ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Negative flow resistance coefficient at output end	Specify resistance coefficient ξ_{out-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end.

19.4.2. Initial Conditions

Initial conditions cannot be specified for **Round-edged Orifice** element.

19.4.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

cross-section area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-section area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---

19.4.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

static pressure before throttle (input end)	Corresponds to pressure p_1: refer to Figure 170.
static pressure after throttle (output end)	Corresponds to pressure p_2 refer to Figure 170.
volumetric flow rate at orifice input end	Fluid flow velocity is calculated by Equation 338 in Section 19.1.6 and multiplied with cross-sectional area at input end A_{in}.
volumetric flow rate at orifice input end	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog.
cumulative flow rate at orifice input end	Cumulative flow rate per calculation range.
flow resistance coefficient in throttle	Internally calculated flow resistance coefficient.
volumetric flow rate at orifice output end	Fluid flow velocity is calculated by Equation 337 in Section 19.1.6 and multiplied with cross-sectional area at output end A_{out}.
volumetric flow rate at orifice output end	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow rate at orifice output end	Cumulative flow rate per calculation range.

19.4.5. Flow Equations

The BOOST Hydsim model of **Round-edged Orifice** is analogous to the model of **Standard Orifice** (refer to Chapter 19.1) except that the flow resistance coefficient ξ_{thr} is calculated as described in Section 19.3.6.

19.4.6. Estimation of Flow Resistance Coefficient

The flow resistance coefficient in the throttle ξ_{thr} is calculated from the flow resistance coefficient in the tube ξ_{nom} by the following formula:

$$\xi_{thr} = \xi_{nom} \frac{d_{thr}^4}{D_{nom}^4}. \tag{349}$$

Flow resistance coefficient in the tube ξ_{nom} is calculated by multiplying the tabular flow resistance coefficient ξ_{tab} with the two correction factors C_{Re} and C_0:

$$\xi_{nom} = \xi_{tab} C_{\mathrm{Re}} C_0. \tag{350}$$

Correction factor C_{Re} depends on the Reynolds number (refer to Section 19.3.6).

Correction factor C_0 for the round-edged orifice is obtained from the diagram in Figure 17**133**.

Flow resistance coefficient ξ_{tab} for the round-edged orifice can be obtained from the Table 11. For calculation, the length of orifice l in the table has to be set to 0.

Table 11: Thick-edged Orifice in Straight Tube

ξ_{nom}		A_0/A_1															
l/D_h	τ	0,02	0,04	0,06	0,08	0,10	0,15	0,20	0,25	0,30	0,40	0,50	0,60	0,70	0,80	0,90	1,0
0	1,35	7000	1670	730	400	245	96,0	51,5	30,0	18,2	8,25	4,00	2,00	0,97	0,42	0,13	0
0,2	1,22	6600	1600	687	374	230	94,0	48,0	28,0	17,4	7,70	3,75	1,87	0,91	0,40	0,13	0,01
0,4	1,10	6310	1530	660	356	221	89,0	46,0	26,5	16,6	7,40	3,60	1,80	0,88	0,39	0,13	0,01
0,6	0,84	5700	1380	590	322	199	81,0	42,0	24,0	15,0	6,60	3,20	1,60	0,80	0,36	0,12	0,01
0,8	0,42	4680	1130	486	264	164	66,0	34,0	19,6	12,2	5,50	2,70	1,34	0,66	0,31	0,11	0,02
1,0	0,24	4260	1030	443	240	149	60,0	31,0	17,8	11,1	5,00	2,40	1,20	0,61	0,29	0,11	0,02
1,4	0,10	3930	950	408	221	137	55,6	28,4	16,4	10,3	4,60	2,25	1,15	0,58	0,28	0,11	0,03
2,0	0,02	3770	910	391	212	134	53,0	27,4	15,8	9,90	4,40	2,20	1,13	0,58	0,28	0,12	0,04
3,0	0	3765	913	392	214	132	53,5	27,5	15,9	10,0	4,50	2,24	1,17	0,61	0,31	0,15	0,06
4,0	0	3775	930	400	215	132	53,8	27,7	16,2	10,0	4,60	2,25	1,20	0,64	0,35	0,16	0,08
5,0	0	3850	936	400	220	133	55,5	28,5	16,5	10,5	4,75	2,40	1,28	0,69	0,37	0,20	0,10
6,0	0	3870	940	400	222	133	55,8	28,5	16,6	10,5	4,80	2,42	1,32	0,70	0,40	0,21	0,12
7,0	0	4000	950	405	230	135	55,9	29,0	17,0	10,9	5,00	2,50	1,38	0,74	0,43	0,23	0,14
8,0	0	4000	965	410	236	137	56,0	30,0	17,2	11,2	5,10	2,58	1,45	0,78	0,45	0,25	0,16
9,0	0	4080	985	420	240	140	57,0	30,0	17,4	11,4	5,30	2,62	1,50	0,80	0,50	0,28	0,18
10	0	4110	1000	430	245	146	59,7	31,0	18,2	11,5	5,40	2,80	1,57	0,89	0,53	0,32	0,20

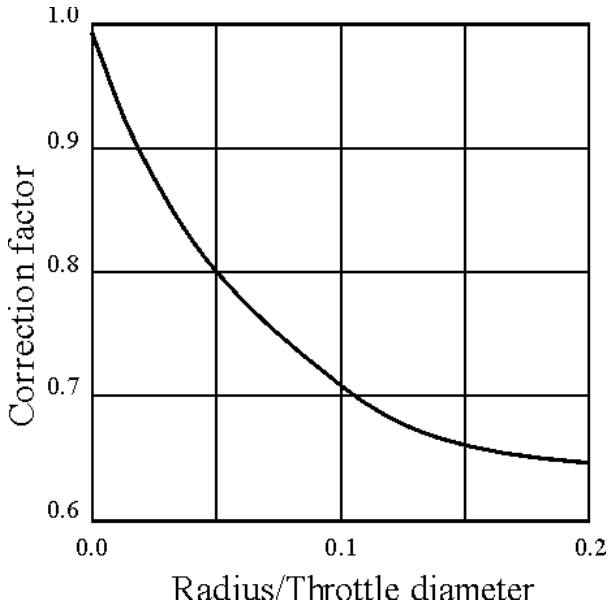
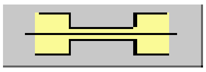
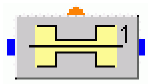


Figure 171: Correction Factor C_0 for Edge Radii of Rounded Orifices

19.5. Sharp-edged Long Orifice

Element Name:	**Sharp-edged Long Orifice**	
Element Icon:		
Definition:	This element serves to define a Sharp-edged Long Orifice.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Schematic of the **Sharp-edged Long Orifice** with expansions/contractions at connections is shown in Figure 172.

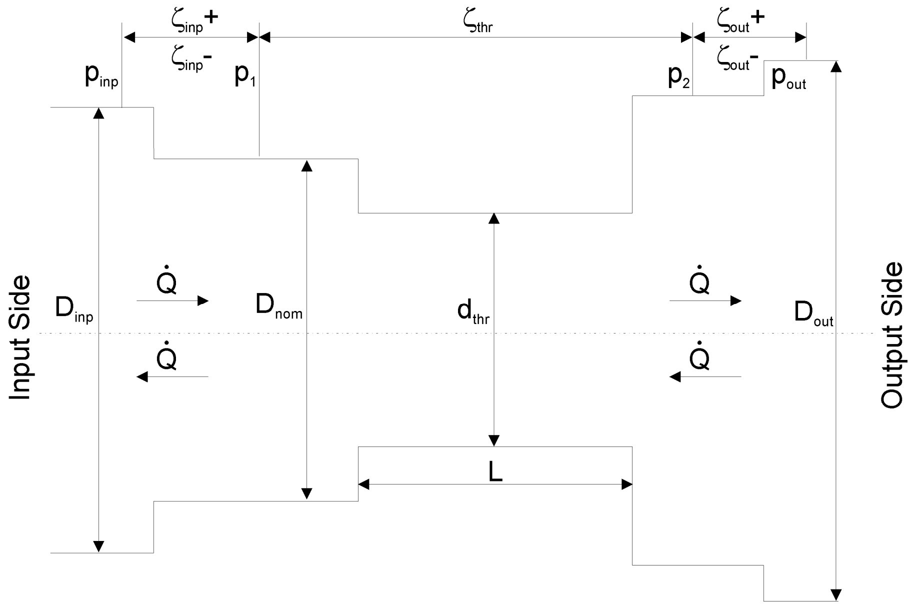


Figure 172: Sharp-edged Long Orifice

19.5.1. Input Parameters

Description of input data for the **Sharp-edged Long Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Tube diameter	Specify tube diameter of Orifice D_{nom}.
Throttle diameter	Specify diameter of throttle D_{thr}.
Throttle length	Specify length of throttle *L*.
EXPANSIONS/ CONTRACTIONS AT CONNECTIONS	
Cross-sectional area at input end	Specify cross-sectional area at input end ($\frac{1}{4}D_{in}^2\pi$).
Cross-sectional area at output end	Specify cross-sectional area at output end ($\frac{1}{4}D_{out}^2\pi$).
Positive flow resistance coefficient at input end	Specify resistance coefficient ξ_{int+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Positive flow resistance coefficient at output end	Specify resistance coefficient ξ_{out+} for positive flow (from input end to output end) related to area A_{nom} at sudden change of cross-sectional area on output end.
Negative flow resistance coefficient at input end	Specify resistance coefficient ξ_{in-} for negative flow (from output end to input end) related to area A_{nom} at sudden change of cross-sectional area on input end.
Negative flow	Specify resistance coefficient ξ_{out-} for negative flow (from

resistance coefficient at output end	output end to input end) related to area A_{nom} at sudden change of cross-sectional area on output end.

19.5.2. Initial Conditions

Initial conditions cannot be specified for **Sharp-edged Long Orifice** element.

19.5.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

cross-sectional area at input connection	SI Unit:	m^2
resistance coefficient at input for positive flow	SI Unit:	---
resistance coefficient at input for negative flow	SI Unit:	---
cross-sectional area at output connection	SI Unit:	m^2
resistance coefficient at output for positive flow	SI Unit:	---
resistance coefficient at output for negative flow	SI Unit:	---

19.5.4. Output Parameters

Path: **Element**| **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

static pressure before throttle (input end)	Corresponds to pressure p_1 (refer to Figure 172)
static pressure after throttle (output end)	Corresponds to pressure p_2 (refer to Figure 172)
volumetric flow rate at orifice input end	Fluid flow velocity is calculated by Equation 338 in Section 19.1.6 and multiplied with cross-sectional area at input end A_{in}.
volumetric flow rate at orifice input end	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog.
cumulative flow rate at orifice input end	Cumulative flow rate per calculation range.
flow resistance coefficient in throttle	Internally calculated flow resistance coefficient.
volumetric flow rate at orifice output end	Fluid flow velocity is calculated by Equation 337 in Section 19.1.6 and multiplied with cross-sectional area at output end A_{out}.

volumetric flow rate at orifice output end	Output values will be 0 if the **Time** domain button is active in **Calculation Control** dialog.
cumulative flow rate at orifice output end	Cumulative flow rate per calculation range.

19.5.5. Flow Equations

The BOOST Hydsim model of **Sharp-edged Long Orifice** is analog to the model of **Standard Orifice** (refer to Chapter 19.1) except that the flow resistance coefficient ξ_{thr} is calculated as described in Section 19.3.6.

19.5.6. Estimation of Flow Resistance Coefficient

The flow resistance coefficient in the throttle ξ_{thr} is calculated from the flow resistance coefficient in the tube ξ_{nom} by the following formula:

$$\xi_{thr} = \xi_{nom} \frac{d_{thr}^4}{D_{nom}^4} . \qquad (351)$$

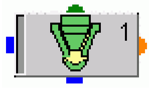
Flow resistance coefficient in the tube ξ_{nom} is calculated by multiplying the tabular flow resistance coefficient ξ_{tab} with the correction factor C_{Re}:

$$\xi_{nom} = \xi_{tab} C_{\mathrm{Re}} . \qquad (352)$$

Flow resistance coefficient ξ_{tab} for the sharp-edged long orifice can be obtained from the **Table 11**. Correction factor C_{Re} depends on the flow regime (Reynolds number). Both coefficients ξ_{tab} and C_{Re} are described in Section 19.3.6.

20. NOZZLE

20.1. Basic SAC Nozzle Orifice

Element Name:	**Basic SAC Nozzle Orifice**	
Element Icon:		
Definition:	This element serves to define the properties of a SAC Nozzle Orifice (basic model with turbulent/cavitating flow).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Note: **Basic SAC Nozzle Orifice** has to be connected to one of the **Needle** elements by a special connection.

Note: **Basic SAC Nozzle Orifice** can use either an external flow model (User-defined Flow Area or Rate) or an internal flow model (Program-calculated Flow Area). Internal BOOST Hydsim model uses same (constant or variable) flow discharge coefficient at needle seat and nozzle spray holes. User-defined flow model requires the input of effective flow area or nozzle flow rate as a function of needle lift.

Note: Hydraulic connections to **SAC Nozzle Orifice** are irreversible: input connection implies a connection to nozzle volume (high pressure volume under needle guide) and output connection is the connection to the spray volume (e.g. combustion chamber, usually modeled as **Boundary** condition).

Note: Flow discharge in **Basic SAC Nozzle Orifice** can be constant or variable (pressure-dependent). Constant flow discharge can be defined by entering the coefficient value or the flow rate and pressure difference across the nozzle. In this case the discharge coefficient is calculated in GUI window.

The cross-section of a **SAC Nozzle Orifice** with spray holes and needle seat is shown in Figure 173.

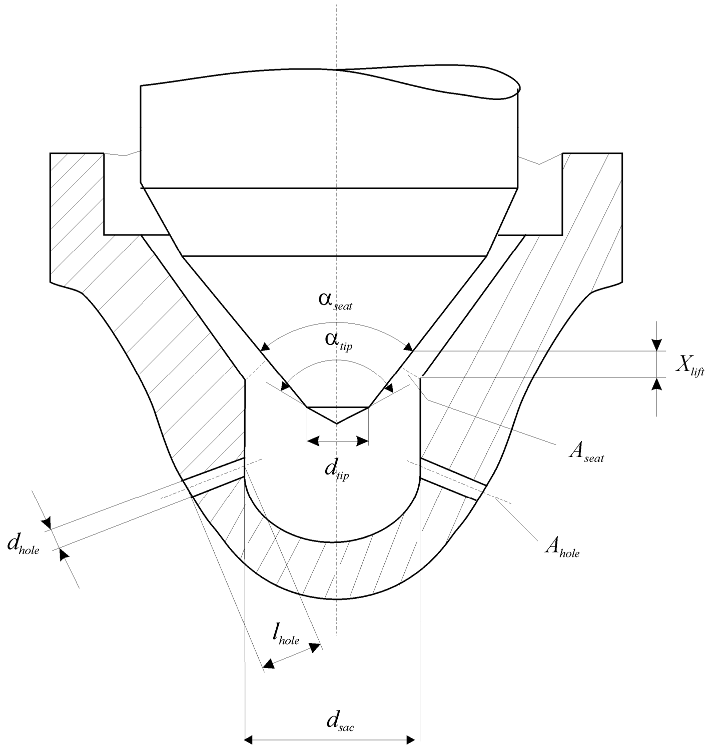


Figure 173: SAC Nozzle Geometry

20.1.1. Input Parameters

Description of input data for the **Basic SAC Nozzle Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Spray Calculation	Press this bar to choose formulae for spray calculation (refer to Section 20.1.4). **Note:** If **Export to FIRE** is active, Spray cone angle calculation has to be activated, too.
Number of spray holes	Specify number of spray holes of nozzle sac N_{holes}.
Diameter of one spray hole	Specify diameter of one spray hole d_{hole} (refer to Figure 173).
Length of one spray hole	Specify length of one spray hole l_{hole} (refer to Figure 173). **Note:** Spray hole length is automatically enabled if the Spray cone angle calculation by **Sitkei** or Spray penetration calculation by **Varde, Popa** is activated.
Diameter of nozzle sac	Specify diameter of nozzle sac d_{sac} (refer to Figure 173).
Angle of needle seat	Specify angle of needle seat α_{seat} (refer to Figure 173).

PROGRAM-CALCULATED FLOW AREA	If this button is active, the internal flow model through nozzle orifice will be used.
<u>Flow Coefficient</u>	For this option flow discharge coefficient has to be defined.
Constant	
Flow discharge coefficient (0<my<1)	Specify turbulent flow discharge coefficient μ..
Critical cavitation number	Specify critical cavitation number ( ) for transition between turbulent and cavitating flow.
Variable	
Discharge Coefficient Table	Press this bar to specify discharge coefficient as function of pressure drop across nozzle.
	Specify pressure difference in ascending order in the first column. Specify flow discharge coefficient in the second column. For pressure difference higher than the highest value specified, the highest pressure drop value from the table is used. For pressure difference lower than the lowest value specified, the lowest pressure drop value from the table is taken (kept constant as long as pressure drop is out of table range). $\Delta p = p_{inp} - p_{out} \quad for \quad p_{inp} \geq p_{out}$ $\Delta p = p_{out} - p_{inp} \quad for \quad p_{inp} < p_{out}$
<u>Flow Rate</u>	For this option flow discharge coefficient is calculated from Bernoulli equation: $\mu = \frac{Q_m}{A}\sqrt{\frac{\rho}{2\Delta p_m}} = \frac{4Q_m}{n\pi d_h^2}\sqrt{\frac{\rho}{2\Delta p_m}}$
Calculate coefficient	Press to calculate flow discharge coefficient and display it at *my* position.
Nozzle flow at max. needle lift	Specify flow rate through nozzle at maximum needle lift Q_m.
USER-DEFINED FLOW AREA OR RATE	Check to use a user-defined flow area or rate (external flow model) instead of program-calculated flow area.
<u>Effective Cross-sectional Flow Area</u>	Click to activate table of effective cross-sectional flow area as a function of needle lift.
<u>Volumetric Flow (Nozzle Rate)</u>	Click to activate table of volumetric flow as a function of needle lift.
Pressure difference	Specify pressure difference Δp_m at which volumetric flow is measured or obtained otherwise.

<table>
<tr><td colspan="2">Volumetric flow $\dot{Q}$ will be internally recalculated into Effective Cross-sectional Flow Area μ according to formula:
$$\mu A = \dot{Q}\sqrt{\frac{\rho}{2\Delta p}}\ , \qquad (353)$$
where Δp is pressure difference defined above and ρ is fluid density.</td></tr>
<tr><td>Factor for 2nd column</td><td>Press to activate the multiplication factor for 2nd column of table.</td></tr>
<tr><td></td><td>Values in the second column will be multiplied with scale factor as soon as Run / Run Sets / Restart is pressed in Simulation menu.
It is convenient to use 'Factor for 2nd column' as discharge coefficient (if it is constant and seat closing/opening areas are available in absolute values not pre-multiplied by μ).</td></tr>
<tr><td>FLOW AREA OR RATE TABLE</td><td>Specify lift of needle in ascending order in the first column. Specify effective cross-sectional flow area (or directly area if μ is used as a 'Factor for 2nd column') or volumetric flow rate through nozzle in the second column.
For needle lifts higher than the highest value specified, the highest μA or $\dot{Q}$ value from the table is used. For lifts lower than the lowest value specified, the lowest μA or $\dot{Q}$ value from the table is taken (kept constant as long as needle lift is out of table range).</td></tr>
</table>

20.1.2. Initial Conditions

Initial conditions cannot be specified for **Basic SAC Nozzle Orifice**.

20.1.3. Modifiable Parameters

Path: **Element**|**Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameter:

flow discharge coefficient	SI Unit:	---

Modification of flow discharge coefficient μ is relevant only if **Program-Calculated Flow Area** is active in Input Data Dialog (internal flow model).

20.1.4. Spray Calculation

This dialog serves to choose empirical formulas for the **Droplet Size Calculation**, **Spray Cone Angle Calculation** and **Spray Penetration Calculation**[34].

> ☞ **Note:** Spray calculation formulas are empirical and derived for the cylindrical spray holes. For the conical holes they may not be applicable.

Description of input data:

Droplet size calculation	Specify formula for the calculation of Sauter mean diameter. • (none) • Knight $d_{32} = \dfrac{1.605 \nu_D^{0.215} \dot{m}^{0.209} \left(\dfrac{A}{A_p} \right)^{0.916}}{\Delta p^{0.458}}$ (354) • Elktob $d_{32} = \dfrac{3.08 \nu_D^{0.385} \sigma_D^{0.737} \rho_D^{0.737} \rho_G^{0.06}}{\Delta p^{0.54}}$ (355) • Varde, Popa $d_{32} = \dfrac{12.392 d_{hole}^{0.44} \rho_D^{0.42} \sigma_D^{0.28} \nu_D^{0.28}}{\Delta p^{0.42} \rho_G^{0.28}}$ (356) • Hiroyasu, Arai $\dfrac{d_{32\_IS}}{d_{hole}} = 4.12\, Re^{0.12}\, We^{-0.75} \left(\dfrac{\mu_D}{\mu_G} \right)^{0.54} \left(\dfrac{\rho_D}{\rho_G} \right)^{0.18}$ $\dfrac{d_{32\_CP}}{d_{hole}} = 0.38\, Re^{0.25}\, We^{-0.32} \left(\dfrac{\mu_D}{\mu_G} \right)^{0.37} \left(\dfrac{\rho_D}{\rho_G} \right)^{-0.47}$ $\dfrac{d_{32}}{d_{hole}} = MAX \left(\dfrac{d_{32\_IS}}{d_{hole}}, \dfrac{d_{32\_CS}}{d_{hole}} \right)$ (357)

[34] Herzog, Peter L. *Möglichkeiten, Grenzen und Vorausberechnung der einspritzspezifischen Gemischbildung bei schnelllaufenden Dieselmotoren mit direkter luftverteilender Kraftstoffeinspritzung*. Fortschr.-Ber. VDI, Reihe 12 Nr. 127. Düsseldorf: VDI-Verlag, 1989.

<table>
<tr><td>Spray cone angle calculation</td><td>
Specify formula for the spray cone angle calculation.
(refer to Figure 174)

(none)
Sitkei

$$\alpha = 0.03824\,\frac{d_{hole}\rho_G^{0.1}\Delta p^{0.35}}{l_{hole}^{0.3}\rho_D^{0.45}\nu_D^{0.7}} \quad (358)$$

Arai, Tabata, Hyroyasu, Shimizu

$$\alpha = 0.05\left(\frac{\Delta p d_{hole}^2}{\rho_G \nu_G^2}\right)^{0.25} \quad (359)$$

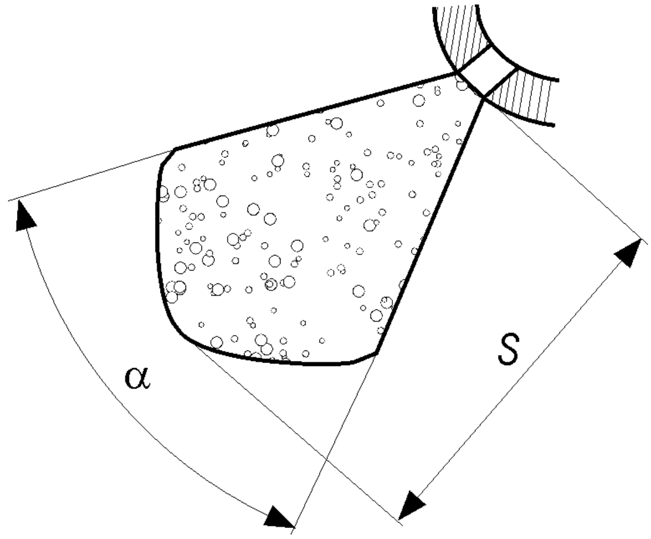


Figure 174: Spray cone angle α and penetration S

Reitz, Bracco

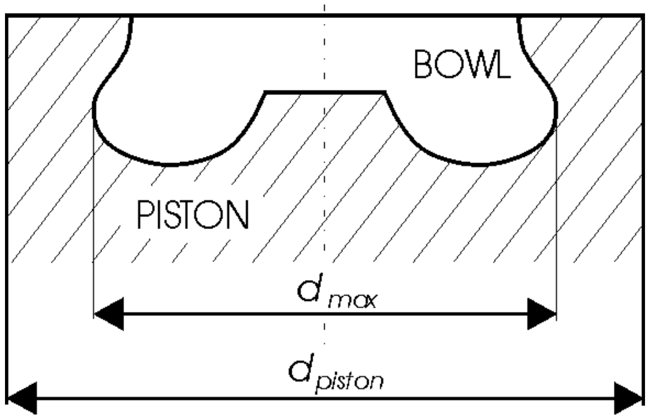
$$\tan\left(\frac{\alpha}{2}\right) = \frac{3.62}{\left(3 + 0.28\,\frac{l_{hole}}{d_{hole}}\right)}\left(\frac{\rho_G}{\rho_D}\right)^{0.5} \quad (360)$$

Shimizu, Hiroyasu

$$\alpha = 83.5\left(\frac{l_{hole}}{d_{hole}}\right)^{-0.22}\left(\frac{d_{hole}}{d_{sac}}\right)^{0.15}\left(\frac{\rho_G}{\rho_D}\right)^{0.26} \quad (361)$$
</td></tr>
<tr><td></td><td>
IMH Munich

$$\alpha = \frac{320\Delta p^{0.15}\rho_G^{0.25}}{\rho_D^{0.45}\nu^{0.1}\left(3 + 0.28\,\frac{l_{hole}}{d_{hole}}\right)\mu_{coeff}^{0.2}} \quad (362)$$
</td></tr>
</table>

Penetration Calculation	Specify formula for the penetration calculation. (refer to Figure 174) • (none) • Lustgarten $$S = 1.593 \frac{d_{hole}^{0.4} \Delta p^{0.27} t^{0.54}}{\rho_D^{0.04} \rho_G^{0.23}} \quad (363)$$ • Varde, Popa $$A_1 = \frac{\Delta p \rho_D d_{hole}^2}{(\nu_D \rho_D)^2}$$ $$A_2 = \frac{\rho_D \sigma_D d_{hole}}{(\nu_D \rho_D)^2}$$ $$A_3 = \frac{\rho_D}{\rho_G}$$ $$A_4 = \frac{l_{hole}}{d_{hole}}$$ $$S = 1.1 d_{hole} A_1^{0.3} A_2^{-0.008} A_3^{0.5} A_4^{0.16} t^{0.55} \quad (364)$$ • Arai, Tabata , Hyroyasu $$t_B = 28.65 \frac{\rho_D d_{hole}}{(\rho_G \Delta p)^{0.5}}$$ if $t < t_B$ $$S = 0.39 t \sqrt{\frac{2}{\rho_D} \Delta p}$$ if $t \geq t_B$ $$S = 2.95 \left(\frac{\Delta p}{\rho_G} \right)^{0.25} \sqrt{d_{hole} t} \quad (365)$$
	• Wakuri $$S = \sqrt{\frac{t d_{hole} \sqrt{\frac{2}{\rho_G} \xi \Delta p}}{0.427 \left(\frac{\rho_G}{\rho_D} \right)^{0.35}}} \quad (366)$$

Inbowl swirl	Specify inbowl swirl which can be calculated from cylinder swirl according to the formula: $$D_{bowl} = D_{cyl.}\left(\frac{d_{piston}}{d_{\max}}\right)^{a} \quad (367)$$ where D_{bowl} is bowl swirl, $D_{cyl.}$ is cylinder swirl and a is exponent which may be taken 1.35 for the retracted combustion chamber and 1.55 for the open combustion chamber.  **Figure 175: Piston bowl**

The parameters are in the above formulas are defined as:

GAS

ρ_G - gas density

ν_G - kinematic viscosity of gas

μ_G - dynamic viscosity of gas

LIQUID (FUEL)

ρ_D - liquid density

ν_D - kinematic viscosity of liquid

μ_D - dynamic viscosity of liquid

σ_D - surface tension

NOZZLE GEOMETRY

d_{hole} - hole diameter

l_{hole} - hole length

$\frac{A}{A_p}$ - areas ratio $\left(\frac{A}{A_p} = 1\right)$

NOZZLE PROPERTIES	
$\dot{m}$	- mass flow rate
Δp	- pressure difference
ξ	- resistance coefficient
μA	- effective flow area
Q	- flow rate
CYLINDER	
D_{bowl}	- inbowl swirl
GENERAL	
t	- time from injection start
n_{ref}	- reference speed
Re	- Reynolds number
We	- Weber number
d_{32_IS}	- droplet size at incomplete spray
d_{32_CS}	- droplet size at complete spray
μ_{coeff}	- nozzle discharge coefficient

20.1.5. Output Parameters

Path: **Element**| **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

volumetric flow rate through nozzle orifice /time	Flow rate through nozzle orifice in time domain calculated from Equation 371 as given in 20.1.6.
volumetric flow rate through nozzle orifice /angle	Flow rate through nozzle orifice in rerefence angle domain. Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow through nozzle orifice	Cumulative flow rate is injection quantity per calculation range.
effective cross-sectional nozzle flow area	Minimal geometric cross-sectional area multiplied by flow discharge coefficient (corresponds to μA).
mass flow rate through nozzle orifice /time	Mass flow rate through nozzle orifice in time domain calculated from the equation: $$m = Q\rho_D . \qquad (368)$$

mass flow rate through nozzle orifice /angle	Mass rate through nozzle orifice in rerefence angle domain. Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative mass flow through nozzle orifice	Cumulative mass rate is injected mass per calculation range.
spray penetration	(refer to Section 20.1.4)
spray penetration with swirl correction	Spray penetration with swirl correction is calculated by the formula: $$S_D = \frac{S}{1 + \frac{0.074 D_{bowl} n_{ref} \rho_D^{0.5}}{\Delta p^{0.5}}} \quad (369)$$
injection power	Injection power is calculated by the formula: $$P_{injection} = \frac{\rho_D Q^3}{2(\mu A)^2} \quad (370)$$
pressure in nozzle sac	Pressure in nozzle sac is calculated from Equation 377.

20.1.6. Flow Equation

Volumetric flow rate (injection rate) through **Basic SAC Nozzle Orifice** is calculated from Bernoulli equation:

$$\dot{Q}_{hyd} = sign(p_{in} - p_{out}) \sqrt{\frac{1}{\frac{1}{\mu_{seat}^2 A_{seat}^2} + \frac{1}{\mu_{hole}^2 A_{holes}^2}}} \sqrt{\frac{2}{\rho} |p_{in} - p_{out}|}, \quad (371)$$

where $\dot{Q}$ is the injection rate, μ_{seat} and μ_{hole} are the flow discharge coefficients at needle seat and spray hole, respectively, A_{seat} and A_{holes} are the open cross-sectional flow areas of needle seat and spray holes, respectively, p_{in} and p_{out} are the pressures on input and output end and ρ is the fluid density. Actually, p_{in} is the pressure in nozzle volume while p_{out} is the pressure in the injection chamber. For direct injection engines, p_{out} is the cylinder pressure.

Basic SAC Nozzle Orifice model assumes that the flow discharge coefficients at needle seat and spray hole are equal: $\mu_{seat} = \mu_{hole} = \mu$. This assumption is justifiable if no external information about these coefficients is available (e.g. from nozzle flow measurements at various needle lifts). In this case equation (371) reduces to

$$\dot{Q}_{hyd} = sign(p_{in} - p_{out}) \mu A \sqrt{\frac{2}{\rho} |p_{in} - p_{out}|}, \quad (372)$$

where $A = \sqrt{\frac{1}{\frac{1}{A_{seat}^2} + \frac{1}{A_{holes}^2}}}$.

For the external flow model through nozzle orifice (option "User-defined Flow Area or Rate"), effective cross-sectional flow area μA or flow rate is directly taken from the input table according to the actual needle lift. For the internal flow model (option "Program-calculated Flow Area"), the narrowest cross-sectional flow area of needle seat A_{seat} is calculated from the following equation:

$$\begin{aligned}
&\forall\, x_{lift} \le 0 : A_{seat} = 0, \\
&\forall\, x_{lift} > 0 : A_{seat} = \pi x_{lift} \sin\frac{\alpha_{seat}}{2}\left(d_{sac} - x_{lift}\sin\frac{\alpha_{seat}}{2}\cos\frac{\alpha_{seat}}{2}\right), \\
&\forall\, A_{seat} > A_{sac} : A_{seat} = A_{sac} = \frac{\pi d_{sac}^2}{4},
\end{aligned} \tag{373}$$

where x_{lift} is the needle lift, α_{seat} is the angle of needle seat and A_{sac} and d_{sac} are the area and diameter of nozzle sac, respectively. Note that if needle seat area is greater than nozzle sac inlet area (this can happen at high needle lift), then narrowest cross-sectional area A_{seat} is assigned to sac inlet area (invariant of needle lift).

Total area of nozzle spray holes A_{holes} is calculated from the formula:

$$A_{holes} = \frac{\pi}{4} N_{holes} d_{hole}^2, \tag{374}$$

where N_{holes} is the number of spray holes and d_{hole} is the diameter of one hole.

Flow rate through needle seat and spray holes can be separately expressed as follows:

$$\dot{Q}_{seat} = sign(p_{in} - p_{sac})\,\mu A_{seat}\sqrt{\frac{2}{\rho}\left|p_{in} - p_{sac}\right|}, \tag{375}$$

$$\dot{Q}_{holes} = sign(p_{sac} - p_{out})\,\mu A_{holes}\sqrt{\frac{2}{\rho}\left|p_{sac} - p_{out}\right|}. \tag{376}$$

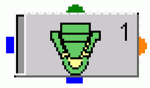
where p_{sac} is the pressure in the nozzle sac volume.

Pressure in the nozzle sac is calculated from the flow continuity condition $\dot{Q}_{seat} = \dot{Q}_{holes}$:

$$\begin{aligned}
&\forall\, x_{lift} \le 0 : \quad p_{sac} = p_{out}, \\
&\forall\, x_{lift} > 0 : \quad p_{sac} = \frac{A_{seat}^2 p_{in} + A_{holes}^2 p_{out}}{A_{seat}^2 + A_{holes}^2}.
\end{aligned} \tag{377}$$

In this way, pressure p_{sac} always stays smaller than pressure p_{in} as long as p_{out} is smaller than p_{in}. Note that for the **Up-to-date Needle** model pressure in the nozzle sac acts under the whole area of needle seat.

20.2. Basic VCO Nozzle Orifice

Element Name:	**Basic VCO Nozzle Orifice**	
Element Icon:		
Definition:	This element serves to define the properties of a Valve-covered Nozzle Orifice (basic model with turbulent/cavitating flow).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connections have to be defined.	

Note: **Basic VCO Nozzle Orifice** has to be connected to one of **Needle** elements by a special connection.

Note: **Basic VCO Nozzle Orifice** can use either an external flow model (User-defined Flow Area or Rate) or an internal flow model (Program-calculated Flow Area). Internal BOOST Hydsim model uses a constant or pressure dependent flow discharge coefficient at needle seat and nozzle spray holes. User-defined flow model requires the input of effective flow area or nozzle flow rate as a function of needle lift.

Note: Hydraulic connections to **VCO Nozzle Orifice** are irreversible: input connection implies a connection to nozzle volume (high pressure volume under needle guide) and output connection is the connection to the spray volume (e.g. combustion chamber, usually modeled as **Boundary** condition).

Note: Flow discharge in **Basic VCO Nozzle Orifice** can be constant or variable (pressure-dependent). Constant flow discharge can be defined by entering the coefficient value or the flow rate and pressure difference across the nozzle. In this case the discharge coefficient is calculated in GUI window.

The cross-section of **VCO Nozzle Orifice** with spray holes and needle seat is shown in Figure 176.

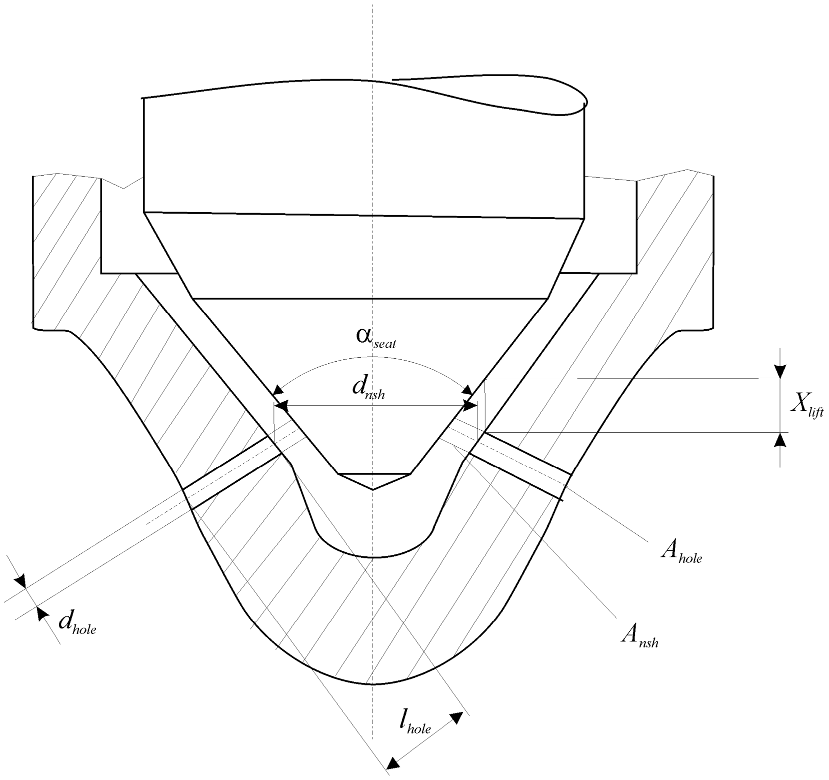


Figure 176: VCO Nozzle Geometry

20.2.1. Input Parameters

Description of input data for the **Basic VCO Nozzle Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Spray Calculation	Press this bar to choose formulae for spray calculation (refer to Section 20.1.4). **Note:** If **Export to FIRE** is active, Spray cone angle calculation has to be activated, too.
Number of spray holes	Specify number of spray holes N_{holes}.
Diameter of one spray hole	Specify diameter of one spray hole d_{hole} (refer to Figure 176).
Length of one spray hole	Specify length of one spray hole l_{hole}. (refer to Figure 176). **Note:** Spray hole length is automatically enabled if the Spray cone angle calculation by **Sitkei** or Spray penetration calculation by **Varde, Popa** is activated.
Nozzle diameter at spray holes	Specify nozzle diameter at spray holes d_{nsh} (refer to Figure 176).
Angle of needle seat	Specify angle of needle seat α_{seat} (refer to Figure 176).
PROGRAM-CALCULATED FLOW AREA	If this button is active, the internal flow model through nozzle orifice will be used (refer to Section 0).

Flow Coefficient	For this option flow discharge coefficient has to be defined.
Constant	
Flow discharge coefficient (0<my<1)	Specify turbulent flow discharge coefficient μ .
Critical cavitation number	Specify critical cavitation number ( ) for transition between turbulent and cavitating flow.
Variable	
Discharge Coefficient Table	Press this bar to specify discharge coefficient as function of pressure drop across nozzle.
	Specify pressure difference in ascending order in the first column. Specify flow discharge coefficient in the second column. For pressure difference higher than the highest value specified, the highest pressure drop value from the table is used. For pressure difference lower than the lowest value specified, the lowest pressure drop value from the table is taken (kept constant as long as pressure drop is out of table range). $\Delta p = p_{inp} - p_{out} \quad for \quad p_{inp} \geq p_{out}$ $\Delta p = p_{out} - p_{inp} \quad for \quad p_{inp} < p_{out}$
Flow Rate	For this option flow discharge coefficient is calculated from Bernoulli equation: $\mu = \frac{Q_m}{A}\sqrt{\frac{\rho}{2\Delta p_m}} = \frac{4Q_m}{n\pi d_h^2}\sqrt{\frac{\rho}{2\Delta p_m}}$
Calculate coefficient	Press to calculate flow discharge coefficient and display it at *my* position.
Nozzle flow at max. needle lift	Specify flow rate through nozzle at maximum needle lift Q_m.
Pressure difference	Specify pressure difference Δp_m at which volumetric flow is measured or obtained otherwise.
Critical cavitation number	Specify critical cavitation number ( ) for transition between turbulent and cavitating flow.
USER-DEFINED FLOW AREA OR RATE	Check to use a user-defined flow area or rate (external flow model) instead of program-calculated area.

Pressure difference	Specify pressure difference Δp_m at which volumetric flow is measured or obtained otherwise.

Volumetric flow $\dot{Q}$ will be internally recalculated into Effective Cross-sectional Flow Area μ according to formula:

$$\mu A = \dot{Q}\sqrt{\frac{\rho}{2\Delta p}}\,, \qquad (378)$$

where Δp is pressure difference defined above and ρ is fluid density.

Factor for 2nd column	Press to activate the multiplication factor for 2nd column of table. Values in second column will be multiplied with scale factor as soon as **Run / Run Sets / Restart** is pressed in **Simulation** menu. It is convenient to use 'Factor for 2nd column' as discharge coefficient (if it is constant and seat closing/opening areas are available in absolute values not premultiplied by μ).
FLOW AREA OR RATE TABLE	Specify lift of needle in ascending order in the first column. Specify effective cross-sectional flow area (or directly area if μ is used as a 'Factor for 2nd column') or volumetric flow rate through nozzle in the second column. For needle lifts higher than the highest value specified, the highest μA or $\dot{Q}$ value from the table is used. For lifts lower than the lowest value specified, the lowest μA or $\dot{Q}$ value from the table is taken (kept constant as long as needle lift is out of table range).

20.2.2. Initial Conditions

Initial conditions cannot be specified for **Basic VCO Nozzle Orifice**.

20.2.3. Modifiable Parameter

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameter:

flow discharge coefficient	SI Unit:	---

Modification of flow discharge coefficient is relevant only if Program-calculated flow area toggle box is active (internal flow model).

20.2.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the button in front of output parameter has to be pressed.

Description of output parameters:

volumetric flow rate through nozzle orifice /time	Flow rate through nozzle orifice in time domain calculated from Equation 382 as given in 0.
volumetric flow rate through nozzle orifice /angle	Flow rate through nozzle orifice in rerefence angle domain. Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow through nozzle orifice	Cumulative flow rate is injection quantity per calculation interval.
effective cross-sectional nozzle flow area	Minimal geometric cross-sectional area multiplied by flow discharge coefficient (corresponds to μA, where area A is obtained from Equation 378).
mass flow rate through nozzle orifice /time	Mass flow rate through nozzle orifice in time domain calculated from the equation: $$m = Q\rho_D \,. \qquad (379)$$
mass flow rate through nozzle orifice /angle	Mass rate through nozzle orifice in rerefence angle domain. Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative mass flow through nozzle orifice	Cumulative mass rate is injected mass per calculation interval.
spray penetration	Refer to Section 20.1.4
spray penetration with swirl correction	Spray penetration with swirl correction is calculated form the formula: $$S_D = \frac{S}{1 + \frac{0.074 D_{bowl} n_{ref} \rho_D^{0.5}}{\Delta p^{0.5}}} \qquad (380)$$
injection power	Injection power is calculated by the formula: $$P_{injection} = \frac{\rho_D Q^3}{2(\mu A)^2} \qquad (381)$$
pressure at spray hole inlet	Pressure at spray hole inlet is calculated from Equation 385.
injection duration	Injection duration is the time interval from injection start till actual time. It stays zero till the first injection starts, then is accumulated with injection time and stays constant as soon as injection stops. When the next injection event (if any) starts, injection duration is reset to zero and calculated in same way again.

time span to next injection	Time span to next injection is the time interval between two subsequent injection events (or initially between calculation start and first injection begin). It is kept constant during injection event and reset to zero at each injection stop. Till next injection start time span is accumulated linearly with real time. Hence at each injection start it reaches a local maximum. Time span to next injection is a counterpart to the injection duration. It is useful for the optimization of time interval between neighboring injection events.
nozzle cavitation number	Cavitation number in the nozzle (narrowest cross-sectional area)
nozzle flow coefficient	Flow coefficient in the nozzle (narrowest cross-sectional area)
mean flow velocity	Average flow velocity (narrowest cross-sectional area)
Reynolds number	Reynolds number value in the nozzle

20.2.5. Flow Equation

Volumetric flow rate (injection rate) through **Basic VCO Nozzle Orifice** is calculated from Bernoulli equation:

$$\dot{Q}_{hyd} = sign(p_{in} - p_{out})\sqrt{\frac{1}{\frac{1}{\mu_{nsh}^2 A_{nsh}^2} + \frac{1}{\mu_{hole}^2 A_{holes}^2}}}\sqrt{\frac{2}{\rho}\left|p_{in} - p_{out}\right|}\,, \tag{382}$$

where $\dot{Q}$ is the injection rate, μ_{nsh} is the flow discharge coefficient in the area before nozzle spray hole, μ_{hole} is the flow discharge coefficient in spray hole, A_{nsh} is the total narrowest flow area before nozzle spray holes, A_{holes} is the open cross-sectional flow areas of spray holes, p_{in} and p_{out} are the pressures on input and output end and ρ is the fluid density. Actually, p_{in} is the pressure in nozzle volume while p_{out} is the pressure in the injection chamber. For direct injection engines, p_{out} is the cylinder pressure.

Basic VCO Nozzle Orifice model assumes that the flow discharge coefficients at needle seat and spray hole are equal: $\mu_{seat} = \mu_{hole} = \mu$. This assumption is justifiable if no external information about these coefficients is available (e.g. from nozzle flow measurements at various needle lifts). In this case equation (382) reduces to

$$\dot{Q}_{hyd} = sign(p_{in} - p_{out})\,\mu A\sqrt{\frac{2}{\rho}\left|p_{in} - p_{out}\right|}\,, \tag{383}$$

where $A = \sqrt{\frac{1}{\frac{1}{A_{nsh}^2} + \frac{1}{A_{holes}^2}}}$.

For the external flow model through nozzle orifice (option „User-defined Flow Area or Rate“), effective cross-sectional flow area μA or flow rate is directly taken from the input table according to the actual needle lift. For the internal flow model (option „Program-calculated Flow Area“) the total narrowest flow area before nozzle spray holes A_{nsh} is calculated as the sum of the areas of quasi-cylindrical surfaces between the needle tip and each spray hole:

$$\begin{aligned} &\forall\, x_{lift} \le 0 : A_{nsh} = 0, \\ &\forall\, x_{lift} > 0 : A_{nsh} = N_{holes} d_{hole} \pi x_{lift} \sin \frac{\alpha_{seat}}{2}, \end{aligned} \tag{384}$$

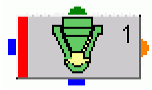
where x_{lift} is the needle lift, α_{seat} is the angle of needle seat, N_{holes} is the number of spray holes and d_{hole} is the diameter of one hole.

Analogously to the **Basic SAC Nozzle** model, pressure under the needle seat (i.e. area before the spray holes) in **Basic VCO Nozzle Orifice** is calculated from the flow continuity condition:

$$\begin{aligned} &\forall\, x_{lift} \le 0 : \quad p_{nsh} = p_{out}, \\ &\forall\, x_{lift} > 0 : \quad p_{nsh} = \frac{A_{nsh}^2 p_{in} + A_{holes}^2 p_{out}}{A_{nsh}^2 + A_{holes}^2}. \end{aligned} \tag{385}$$

In this way, pressure p_{nsh} stays always smaller than p_{in} as long as $p_{out}t$ is smaller than p_{in}.

20.3. Extended SAC Nozzle Orifice

Element Name:	**Extended SAC Nozzle Orifice**	
Element Icon:		
Definition:	This element serves to define the properties of a SAC Nozzle Orifice (extended model with cavitation).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Note: **SAC Nozzle Orifice** is usually used to model the whole nozzle (throttling through needle seat and spray holes). In this case it has to be connected by special connection to a **Needle** element. Via this connection **Nozzle Orifice** receives information about the actual **Needle** lift, and **Needle** element takes information about the pressure at needle seat area etc.

Note: **Extended SAC Nozzle Orifice** can be used to model the throttling through needle seat and spray holes separately. To build up a complete nozzle flow model in this case, two **Extended SAC Nozzle** elements are required. They have to be connected to a **SAC Volume** element. Furthermore, the **Extended SAC Nozzle** element representing the throttling through needle seat has to be connected by special connection to a **Needle** element.

☞ **Note:** For **SAC Nozzle Orifice** connected to a **Needle** element by special connection, the hydraulic connections are irreversible: input connection implies a connection to nozzle volume (high pressure volume under needle guide) and output connection implies the connection to the sac volume or spray (e.g. combustion) chamber.

Note: **Extended SAC Nozzle Orifice** can be used to model the gas inflow into SAC volume (under the needle seat) as a result of higher pressure in cylinder chamber. This may be done in combination with **Gas Boundary** and **Two-phase Volume** elements only (refer to Section 20.3.6).

Note: For **Extended SAC Nozzle Orifice** connected to a **Fire Link** element, the flow through the nozzle and pressure distribution along needle tip is calculated by the FIRE solver (3D multi-phase flow simulation).

The cross-section of a **SAC Nozzle Orifice** with spray holes and needle seat is shown in Figure 173.

20.3.1. Input Parameters

Description of input data for the **Extended SAC Nozzle Orifice**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Spray Calculation	Press this bar to choose formulas for spray calculation (refer to Section 20.1.4). **Note:** If **Export to FIRE** is active, Spray cone angle calculation has to be activated, too.
FLOW THROUGH SPRAY HOLES	Click to activate flow calculation through nozzle spray holes. **Note:** If nozzle sac is modeled as a separate Volume element (e.g. in case of large sac), then two Extended SAC Nozzle elements are necessary (refer to Figure 177). The first element represents the throttling through needle seat (only „Flow through Needle Seat“ option active). It has to be connected by a special connection to the needle, by input hydraulic connection to the nozzle chamber and by output hydraulic connection to the sac volume. The second element represents the flow through nozzle spray holes (only „Nozzle Spray Holes“ option active). It has to be connected by input hydraulic connection to the sac volume and by output hydraulic connection to the spray chamber (boundary pressure), too.

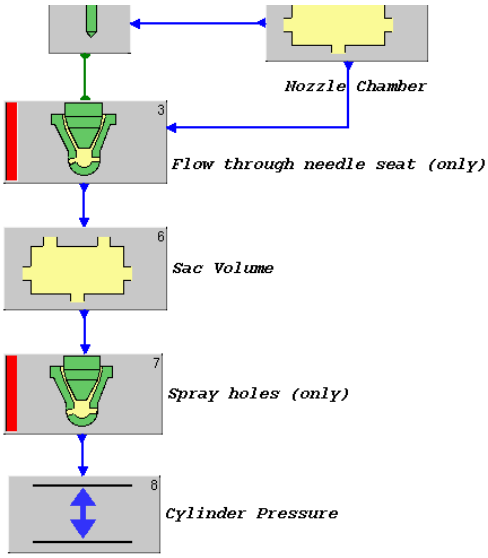


Figure 177: Nozzle Model with a Separate Sac Volume

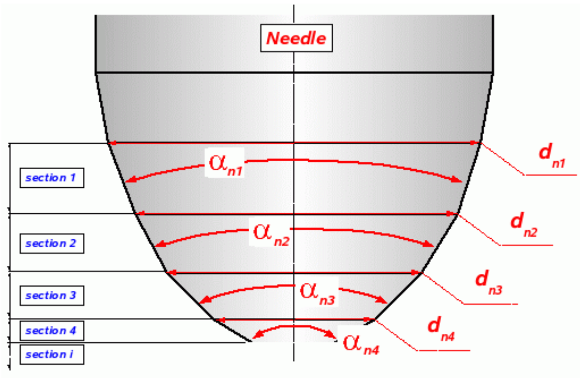
Number of spray holes	Specify the number of spray holes of the nozzle N_{holes}.
Hole outlet diameter	Specify the hole outlet diameter d_{hole}. (refer to Figure 173).
Spray hole length	Specify length of one spray hole l_{hole}. (refer to Figure 173). **Note:** Spray hole length is automatically enabled for the Spray cone angle calculation by **Sitkei** formula, Spray penetration calculation by **Varde, Popa** formula or in case of active „Program-calculated flow discharge coefficient“ option.

User-defined flow discharge coefficient	Click to activate user-defined flow discharge coefficient.
Hole discharge coefficient (no cavitation)	Specify flow discharge coefficient of nozzle spray holes at hydraulic (turbulent) flow μ_{hyd}.
Hole discharge coefficient (at cavitation)	Specify flow discharge coefficient of nozzle spray holes at cavitating flow μ_{cav} (spray hole exit completely filled with the liquid vapor). The cavitation model is described in Chapter 17.2.6 and Egartner[35].
Discharge coefficient at hole inlet	Specify flow discharge coefficient at spray hole inlet μ_{inlet}.
Program-calculated flow discharge coefficient	Click to activate program-calculated flow discharge coefficient (refer to Section 20.3.5.2).
k-factor	Specify *k-factor* for conical holes: $$k - factor = \frac{D_{inlet} - d_{outlet}}{10\mu m}$$ It is active only with „Program-calculated flow discharge coefficient“ option.
Hole inlet radius	Specify inlet radius of the nozzle spray hole r_{inlet}. If hole inlet radius is not defined or set to zero, sharp-edged hole inlet is assumed.
Vapor pressure	Specify fluid vapor pressure in the spray hole p_{vap}.
FLOW THROUGH NEEDLE SEAT	Click to activate flow rate calculation through needle seat. **Note:** It can be activated only **if Extended SAC Nozzle Orifice** is connected to a **Needle** element by a special connection.
Needle Tip/Seat Geometry	
Standard	
Diameter of nozzle sac	Specify diameter of nozzle sac d_{sac}.
Angle of needle seat	Specify angle of needle seat α_{seat}.
Diameter of needle tip	Specify diameter of needle tip d_{tip}.
Angle of needle tip	Specify angle of needle tip α_{tip}.
Multi-section Contour **Input type: Section angle**	

[35] Egartner, Wolfgang. *Parameterschätzung in eindimensionalen Modellen für instationäre Strömung in Dieseleinspritzsystemen*, Dissertation, TU Graz, 1996.

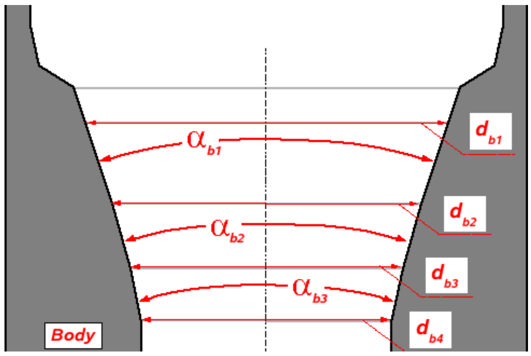
<table>
<tr><td>Needle Tip</td><td>
Needle tip contour table:

- Specify base diameter d_{ni} of truncated cone (frustum) in the 1st column in descending order
- Specify cone angle α_{ni} of tip frustum in the 2nd column in ascending order
- Specify discharge coefficient μ_{ni} at the flow area near respective frustum base (tip contour edge)



</td></tr>
<tr><td>Diameter of needle tip end</td><td>Specify diameter of needle tip end d_{end}.</td></tr>
<tr><td>Nozzle Body</td><td>
Nozzle body contour table:

- Specify base diameter d_{bi} of truncated cone (frustum) in the 1st column in descending order
- Specify cone angle α_{bi} of body frustum in the 2nd column in descending order



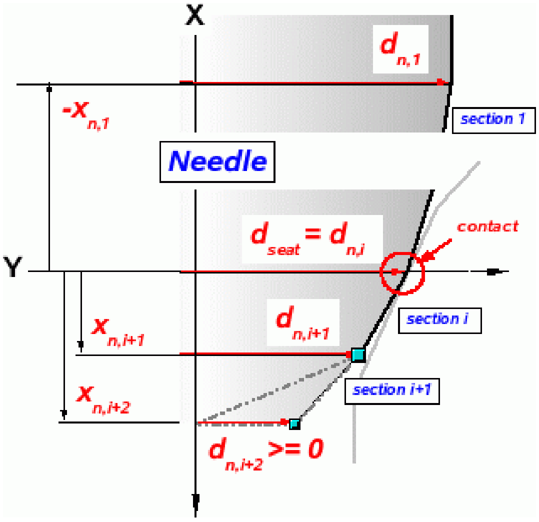
</td></tr>
<tr><td>Diameter of nozzle sac end</td><td>
Specify diameter of nozzle sac end

Note: diameter of nozzle sac end implies the bottom diameter of the nozzle body contour defined in above table.
</td></tr>
<tr><td>Flow discharge at nozzle sac</td><td>Specify flow discharge coefficient at nozzle sac inlet</td></tr>
<tr><td colspan="2"><u>Multi-section Contour</u>

Input type: Base coordinate</td></tr>
</table>

<table>
<tr><td>Needle Tip</td><td>
Needle tip contour table:

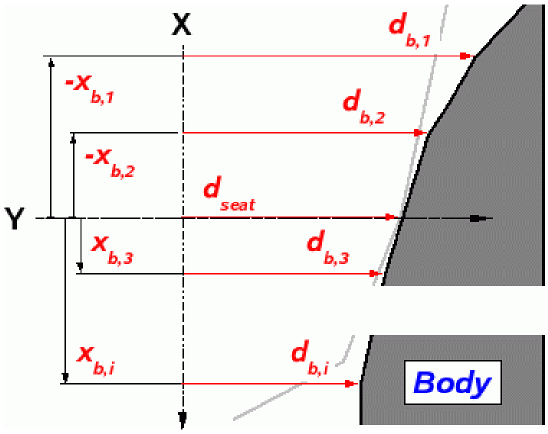
Specify diameter d_{ni} of frustum (truncated cone) base in the 1st column (arbitrary geometry)
Specify x-coordinate x_{ni} of frustum base in the 2nd column in ascending order
Specify discharge coefficient μ_{ni} at the flow area near respective frustum base (tip contour edge)



Notes (input requirements):

Coordinate system origin is the intersection point between the vertical needle symmetry axis x and seat plane (horizontal axis y)
For a sharp needle tip (usual case), base diameter in last row has to bet set to zero
Sign definition rule of base x-coordinate in 2nd column: for base above seat x-coordinate is negative (–x), at needle seat x-coordinate is zero (x=0, this input row must exist) and for base below seat x-coordinate is positive (+x)
Flow coefficient of any effective area inside needle tip section is calculated by the linear interpolation of the flow coefficients between the neighboring table rows (contour edges). This coefficient applies for turbulent flow calculation only
Flow coefficient in last table row has to be the inlet discharge coefficient of nozzle sac

</td></tr>
<tr><td></td><td></td></tr>
</table>

<table>
<tr><td>Nozzle Body</td><td>Nozzle body contour table:
Specify base diameter d_{bi} of truncated cone (frustum) in the 1st column in non-ascending order
Specify x-coordinate x_{bi} of frustum base in the 2nd column in ascending order

Notes (input requirements):
Coordinate system origin is the intersection point between the vertical needle symmetry axis x and seat plane (horizontal axis y)
Nozzle body contour has to be defined till nozzle sac inlet
Base diameter in last table row has to be nozzle sac diameter (bottom diameter of body contour)
Sign definition rule of base x-coordinate in 2nd column: for base above seat x-coordinate is negative (–x) and for base below seat x-coordinate is positive (+x)
Nozzle body contour cannot contain an edge at needle seat (i.e. input x=0 is not allowed)</td></tr>
<tr><td colspan="2"><u>Loss coefficient at needle seat</u></td></tr>
<tr><td>Constant for laminar flow</td><td>Specify loss constant C_{lam} in the numerator of the equation below (also refer to Equation 386 in Section 20.3.5).</td></tr>
<tr><td>Exponent of Reynolds number</td><td>Specify the exponent γ of Reynolds number Re.
Definition of loss coefficient for laminar flow ξ_{lam}:
$$\xi_{lam} = \frac{C_{lam}}{\mathrm{Re}^{\gamma}}$$</td></tr>
<tr><td>Turbulent flow coefficient</td><td>Press this bar to specify turbulent flow coefficient ξ_{turb}. It can be constant value or function of needle lift.
Note: Turbulent flow coefficient is inactive for “Multi-section Contour” option. In this case turbulent flow coefficients are defined in 3rd column of Needle tip contour table.</td></tr>
<tr><td><u>Constant</u></td><td></td></tr>
<tr><td>Turbulent flow constant at needle seat</td><td>Specify loss coefficient for turbulent flow (if appropriate).</td></tr>
</table>

Needle Lift-dependent	• Specify lift of needle in ascending order in the 1st column. • Specify turbulent flow coefficient at opening in the 2nd column. • Specify turbulent flow coefficient at closing in the 3rd column. For needle lifts higher than the highest value specified, the last ξ_{turb} value from the table is used. For lifts lower than the lowest value specified, the first ξ_{turb} value from the table is taken (kept constant as long as needle lift is out of table range).
(*1) **Note:** For SAC Nozzle Orifice connected to a Fire Link element, some parameters are disabled (grayed out) because they are not necessary for a coupled BOOST Hydsim-FIRE calculation.	

20.3.2. Initial Conditions

Initial conditions cannot be specified for **Extended SAC Nozzle Orifice** element.

20.3.3. Modifiable Parameters

Path: **Element**| **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

hole discharge coefficient (no cavitation)	SI Unit:	---
hole discharge coefficient (at cavitation)	SI Unit:	---
constant for laminar flow at needle seat	SI Unit:	---
turbulent flow constant at needle seat	SI Unit:	---
flow discharge coef. at hole inlet	SI Unit:	---

20.3.4. Output Parameters

Path: **Element**| **Store Results**

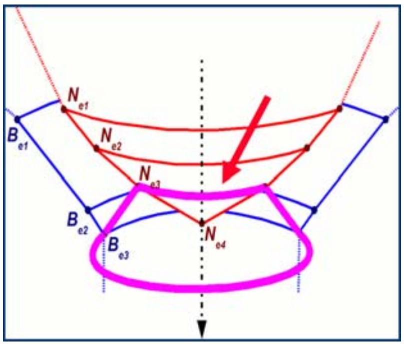
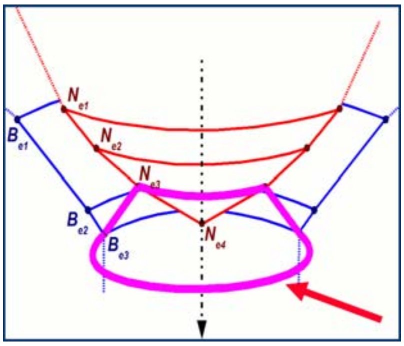
To activate output parameter, the option box in front of output parameter name has to be checked.

Description of output parameters:

volumetric flow rate through nozzle orifice	Volumetric flow rate through nozzle orifice is calculated from Equation 386.
volumetric flow rate through nozzle orifice	Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.

cumulative flow rate through nozzle orifice	Cumulative rate is injection quantity per calculation range.
pressure in nozzle sac	Pressure in sac is calculated according to Equation 395 or 396 located in Section 20.3.5.
geometric flow area at needle seat	Geometric flow area at needle seat cross-section is calculated according to geometrical characteristics of needle seat. It is the minimum shell area of the truncated cone between needle tip and nozzle body surfaces. In case of **Standard Needle Tip/Seat Geometry** it is calculated from Equation 387. For **Multi-section Geometry** area calculation is based on numerical algorithm described in section 20.3.6.1.
cavitation switch	Possible values: 0 – flow (needle closed); 1 – hydraulic flow; 2 – cavitating flow
cavitation factor in nozzle orifice	Cavitation factor is calculated from the relationships in Equations 391 and 392 located in Section 20.3.5.
effective flow area at 'vena contracta'	for turbulent flow: $$A_{vena} = \frac{A_{hole\_i}}{\sqrt{\frac{1}{\mu_{inlet}^2} + \frac{1}{\mu_{hyd}^2}}}$$ for cavitating flow: $$A_{vena} = \mu_{cav} A_{hole\_i}$$

effective flow area at spray hole exit	for turbulent flow: $$A_{eff} = A_{hole\_i}$$ for cavitating flow: $$\mathrm{v}_{mean} = \frac{Q}{A_{hole\_i}}$$ $$\mathrm{v}_{vena} = \sqrt{\frac{2p_{in}}{\rho}}$$ $$\mathrm{v}_{eff} = \mathrm{v}_{vena} - \frac{p_{out}}{\rho \mathrm{v}_{mean}}$$ $$A_{eff} = \frac{\mathrm{v}_{mean}}{\mathrm{v}_{eff}} A_{hole\_i}$$ where: v_{mean} - mean flow velocity v_{vena} - flow velocity in 'vena contracta' v_{eff} - flow velocity in effective exit area A_{hole_i} - cross-sectional area of one spray hole
mass flow rate through nozzle orifice	Mass flow rate through nozzle orifice is a product of volumetric rate and fuel density: $$m = Q\rho_D .$$
mass flow rate through nozzle orifice	Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative mass flow through nozzle orifice	Cumulative mass rate is the injected mass per whole calculation range.
spray calculation ...	Refer to Section 20.1.4
injection duration	Injection duration is the time interval from injection start till actual time. It stays zero till the first injection starts, then is accumulated with injection time and stays constant as soon as injection stops. When the next injection event (if any) starts, injection duration is reset to zero and calculated in same way again.
time span to next injection	Time span to next injection is the time interval between two subsequent injection events (or initially between calculation start and first injection begin). It is kept constant during injection event and reset to zero at each injection stop. Till next injection start time span is accumulated linearly with real time. Hence at each injection start it reaches a local maximum. Time span to next injection is a counterpart to the injection duration. It is useful for the optimization of time interval between neighboring injection events.

flow switch at seat (none/laminar/turbulent)	Flow switch (regime) is determined from to Equation 389. Flow resistance coefficient ξ_{seat} in Equation 386 depends on the flow regime (laminar or turbulent).
effective flow area (frustum shell) at needle seat	This is minimum effective flow area at needle seat cross-section (frustum shell). It is calculated as a product of geometric flow area at needle seat and flow discharge coefficient for this area.
position of geometric frustum base 1 on needle tip contour	This number references the position of geometric frustum base 1 on the needle tip contour. It is calculated by $$C_{Nj} = N_{ej} + \lambda_j, \lambda_j \in [0,1],$$ where N_{ej} is the edge number of needle contour (*j*-th row in Needle Tip table) and λ is the position index of base 1 within the needle tip section $[j, j+1]$. For integer value N_{ei} , base 1 position corresponds to the needle tip contour edge as shown in Figure below. If N_{ej} =0, minimum geometric flow area is the area of nozzle sac. 
position of geometric frustum base 2 on nozzle body contour	This number references the position of geometric frustum base 2 on the nozzle body contour. It is calculated by $$C_{Bi} = B_{ei} + \lambda_i, \lambda_i \in [0,1],$$ where B_{ei} is the edge number of nozzle body contour (*i*-th row in Nozzle Body table) and λ is the position index of base 2 within the nozzle body section $[i, i+1]$. For integer value B_{ei} , base 2 position corresponds to the nozzle body contour edge as shown in Figure below. 

position of effective area cone base 1 on needle tip contour	This number references the position of effective frustum base 1 on the needle tip contour. It is calculated in the same way as for geometric area.
position of effective area cone base 2 on nozzle body contour	This number references the position of geometric frustum base 2 on the nozzle body contour. It is calculated in the same way as for geometric area.

20.3.5. Flow Equations

The flow model through **Extended SAC Nozzle Orifice** is shown in Figure 178. It shows two in series switched orifices which represent the one-dimensional flow through needle seat and nozzle spray holes, respectively.

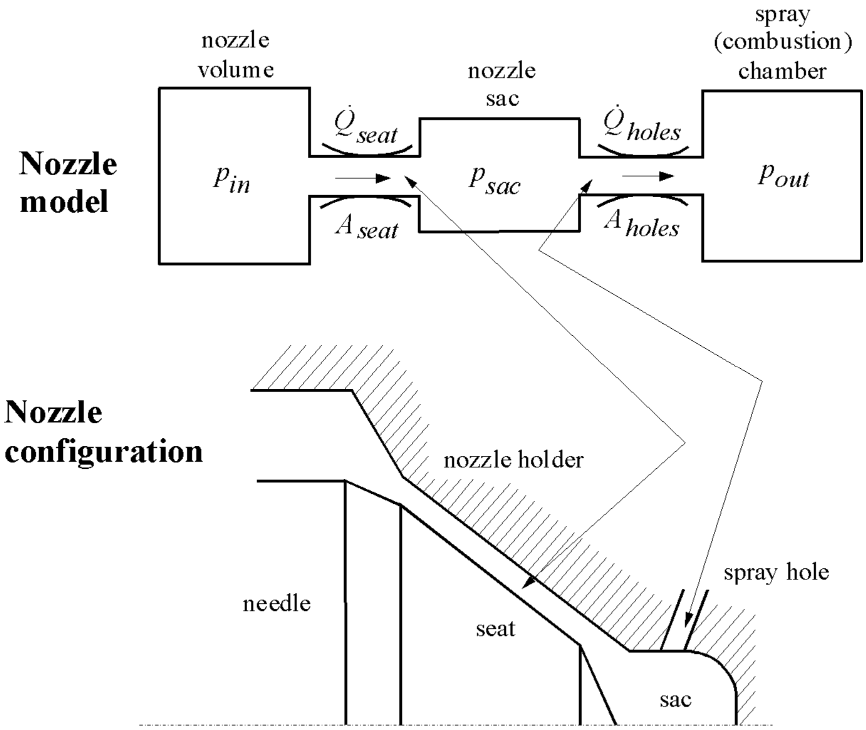


Figure 178: Flow Model through Extended SAC Nozzle Orifice

Volumetric flow rate (injection rate) through **Extended SAC Nozzle Orifice** is calculated from Bernoulli equation:

$$\dot{Q} = sign(p_{in} - p_{out}) \sqrt{\frac{1}{\frac{\xi_{seat}}{A_{seat}^2} + \frac{\xi_{inlet}}{A_{holes}^2} + \frac{\xi_{holes}}{A_{holes}^2}}} \sqrt{\frac{2}{\rho}|p_{in} - p_{out}|}\,, \qquad (386)$$

where $\dot{Q}$ is the injection rate, ξ_{seat}, ξ_{inlet} and ξ_{holes} are the flow resistance coefficients at needle seat, nozzle spray hole inlet and nozzle spray holes, A_{seat} is the narrowest cross-sectional flow area of needle seat, A_{holes} is the total cross-sectional area of nozzle spray holes, p_{in} and p_{out} are the pressures on input and output end and ρ is the fluid density. Actually, p_{in} is the pressure in nozzle volume while p_{out} is the pressure in the injection chamber. For direct injection engines, p_{out} is the cylinder pressure.

Area A_{seat} is equal to the shell area of the smallest truncated cone (frustum) between the needle tip surface and nozzle body surface. It is calculated from the following equation:

$$\begin{aligned}
&\forall\, d_{sac} \le \left(d_{tip} + x \sin\alpha_{seat}\right): \\
&A_{seat} = \pi x_{lift} \sin\frac{\alpha_{seat}}{2}\left(d_{tip} + x_{lift}\sin\frac{\alpha_{seat}}{2}\cos\frac{\alpha_{seat}}{2}\right), \\
&\forall\, d_{sac} > \left(d_{tip} + x \sin\alpha_{seat}\right): \\
&A_{seat} = \pi x_{lift} \sin\frac{\alpha_{seat}}{2}\left(d_{sac} - x_{lift}\sin\frac{\alpha_{seat}}{2}\cos\frac{\alpha_{seat}}{2}\right), \\
&\forall\, A_{seat} > A_{sac} : A_{seat} = A_{sac} = \frac{\pi d_{sac}^2}{4},
\end{aligned} \tag{387}$$

where x_{lift} is the needle lift, α_{seat} is the angle of needle seat, A_{sac} and d_{sac} are the area and diameter of nozzle sac, respectively, and d_{tip} is the diameter of needle tip. Note that if needle seat area is greater than nozzle sac inlet area (this can happen at high needle lift), then narrowest cross-sectional area A_{seat} is assigned to sac inlet area (refer to Figure 173). Needle lift of a **Standard** and **Two-spring Injector Nozzle** is calculated from the differential equations of motion in Chapter 19.

Total area of nozzle spray holes A_{holes} is calculated from the formula:

$$A_{holes} = \frac{\pi}{4} N_{holes} d_{hole}^2 , \tag{388}$$

where N_{holes} is the number of spray holes and d_{hole} is the diameter of one hole.

Flow resistance coefficient ξ_{seat} in Equation 386 depends on the flow type (laminar or turbulent). It is calculated from the following condition:

$$\begin{aligned}
&\forall\, \xi_{lam} > \xi_{turb} :\ \xi_{seat} = \xi_{lam} = \frac{C_{lam}}{\mathrm{Re}^{\gamma}} , \\
&\forall\, \xi_{lam} \le \xi_{turb} :\ \xi_{seat} = \xi_{turb} ,
\end{aligned} \tag{389}$$

where ξ_{lam} and ξ_{turb} are the resistance coefficients for laminar and turbulent flow in the nozzle, C_{lam} is the loss constant for laminar flow, Re is Reynolds number and γ is the exponent. AVL default estimate for C_{lam} is 5000. However, it may vary in a wide range depending on the nozzle type.

20.3.5.1. User-defined flow discharge coefficient

Depending on the flow regime (hydraulic or cavitating) at nozzle spray holes, flow resistance coefficient at spray holes is calculated from:

$$\begin{aligned} &\text{hydraulic}: \ \xi_{holes} = \frac{1}{\mu_{hyd}^2}, \\ &\text{cavitating}: \ \xi_{holes} = \frac{1}{\mu_{cav}^2}, \end{aligned} \tag{390}$$

where μ_{hyd} and μ_{cav} are the flow discharge coefficients at nozzle spray holes for hydraulic and cavitating flow, respectively.

Flow discharge coefficient at cavitation μ_{cav} can be estimated from experiment by measuring the flow transition to cavitation in the nozzle. Cavitation phenomena in nozzle orifices are described in Section 17.2.6 and, more in detail, in reference[36].

Transition to cavitation is determined by the following conditions (refer to inequalities (17.2.13) and (17.2.14) in Section 17.2.6):

$$\frac{|p_{in} - p_{out}|}{p_{in} - p_{vap}} \le \left(\frac{\mu_{cav}}{\mu_{hyd}} \right)^2 \Rightarrow \text{no cavitation,} \tag{391}$$

$$\frac{|p_{in} - p_{out}|}{p_{in} - p_{vap}} > \left(\frac{\mu_{cav}}{\mu_{hyd}} \right)^2 \Rightarrow \quad \text{cavitation,} \tag{392}$$

where p_{vap} is the fuel vapor pressure in the nozzle. Here it is assumed $p_{vap} \approx 0$ because vapor pressure is negligible compared to input pressure in the nozzle volume.

Thus, for the hydraulic flow (no cavitation) in the nozzle, Equation 386 for the calculation of the injection rate can be written in the form:

$$\dot{Q}_{hyd} = sign(p_{in} - p_{out}) \sqrt{\frac{1}{\frac{\xi_{seat}}{A_{seat}^2} + \frac{1}{\mu_{inlet}^2 A_{holes}^2} + \frac{1}{\mu_{hyd}^2 A_{holes}^2}}} \sqrt{\frac{2}{\rho} |p_{in} - p_{out}|} \,. \tag{393}$$

In case of full cavitation in nozzle spray holes, the output pressure for Bernoulli equation is the vapor pressure: $p_{out} = p_{vap} \approx 0$. Furthermore, according to the potential flow theory the losses at hole inlet can be neglected[37]. Hence equation 386 for the cavitating flow reduces to:

[36] Ofner H., Egartner W. *Identification of Flow Phenomena in Fuel Injection Systems for Diesel Engines from Dynamic Measurements*, C499/053, IMechE, 1996.

$$\dot{Q}_{cav} = \sqrt{\frac{1}{\frac{\xi_{seat}}{A_{seat}^2} + \frac{1}{\mu_{cav}^2 A_{holes}^2}}} \sqrt{\frac{2}{\rho} p_{in}} \,. \tag{394}$$

Pressure in the nozzle sac depends on the cavitation condition. For hydraulic flow, satisfying Equation 391, it is calculated from the equation:

$$p_{sac} = p_{out} + \left(\frac{\xi_{inlet}}{A_{holes}^2} + \frac{\xi_{holes}}{A_{holes}^2} \right) \dot{Q}^2 \frac{\rho}{2} \,. \tag{395}$$

For cavitating flow, satisfying Equation 392, pressure in the nozzle sac is given by:

$$p_{sac} = p_{in} - \frac{\xi_{seat}}{A_{seat}^2} \dot{Q}^2 \frac{\rho}{2} . \tag{396}$$

Note that for the **Up-to-date Needle** model pressure in the nozzle sac acts under the whole area of needle seat. Density ρ would be different at the needle seat (Equation 396) and nozzle spray holes (Equation 395) if variable fluid properties are defined for the nozzle.

Hydraulic diameter of the flow pattern (stream tube) through needle seat is given by:

$$D_h = \frac{4 A_{seat}}{P_{seat}} \,, \tag{397}$$

where P_{seat} is the needle seat perimeter at the narrowest cross-sectional flow area. P_{seat} is calculated from:

$$\begin{aligned} &\forall\, d_{sac} \le \left(d_{tip} + x \sin\alpha_{seat}\right):\ P_{seat} = \pi\left(2 d_{tip} + x_{lift} \sin\alpha_{seat}\right), \\ &\forall\, d_{sac} > \left(d_{tip} + x \sin\alpha_{seat}\right):\ P_{seat} = \pi\left(2 d_{sac} - x_{lift} \sin\alpha_{seat}\right). \end{aligned} \tag{398}$$

Reynolds number for the calculation of laminar flow resistance coefficient (refer to Equation 389) is obtained from the formula:

$$\mathrm{Re} = \frac{\left(\frac{\dot{Q}}{A_{seat}} \right) D_h}{\nu} = \frac{4 \dot{Q}}{P_{seat} \nu} \,, \tag{399}$$

where ν is the kinematic viscosity of the fluid.

Obviously, with the increasing flow through needle seat Reynolds number will become larger and flow resistance coefficient ξ_{seat} – smaller. As soon as ξ_{seat} drops down to the prescribed value ξ_{turb}, it will be kept constant at this value. AVL default value for the resistance coefficient ξ_{turb} is 1 because turbulent flow occurs only at higher needle lifts, when the resistance at needle seat becomes small compared to the resistance at nozzle spray holes. The user is free to specify any value of this coefficient (greater than 1) if it can be verified by experimental data or justified otherwise.

20.3.5.2. Program-calculated flow discharge coefficient (internal nozzle flow model)

Internal nozzle flow model is based on the phenomenological flow model in the nozzle holes[37]. The nozzle walls are assumed to be perfectly smooth and the flow field to be axis-symmetric. The chamber before the nozzle is big enough, so that the streamlines at the inlet are not influenced by the walls. Different flow conditions, losses and pressure drop in the nozzle hole are described below:

(a) **Turbulent flow**

Flow losses are caused by the acceleration, formation of the velocity profile at the inlet, expansion after vena contracta and wall friction.

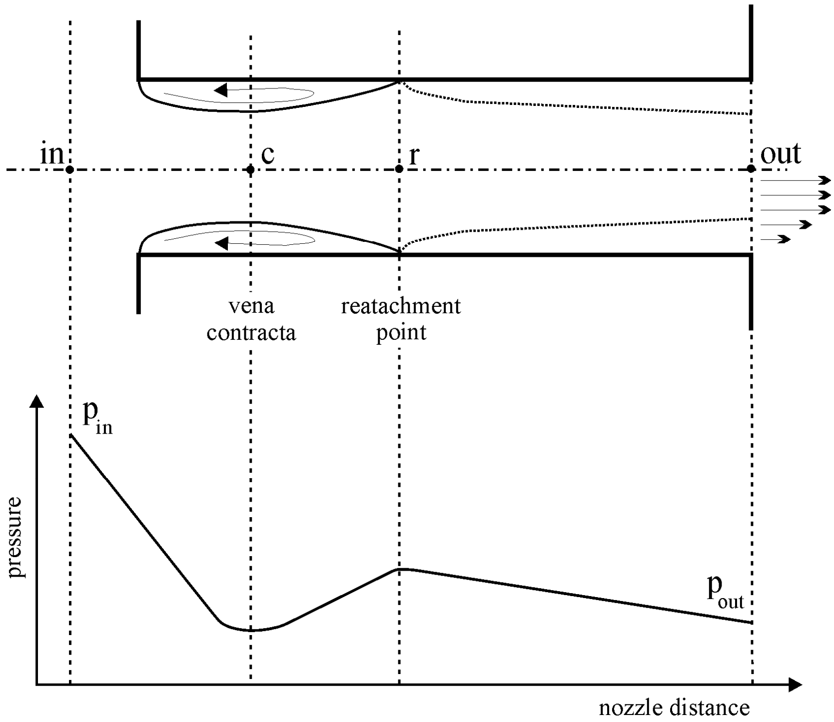


Figure 179: Turbulent Flow in Nozzle Hole

(b) **Onset of Cavitation**

The second regime (onset of cavitation) describes the cavitation that does not reach the nozzle exit. This case usually prevails over a very short range of injection pressures.

[37] Von Kuensberg Sarre, Ch., Kong, S-Ch. and Reitz, R.D. *Modeling the Effects of Injector Nozzle Geometry on Diesel Sprays*. SAE Paper 1999-01-0912.

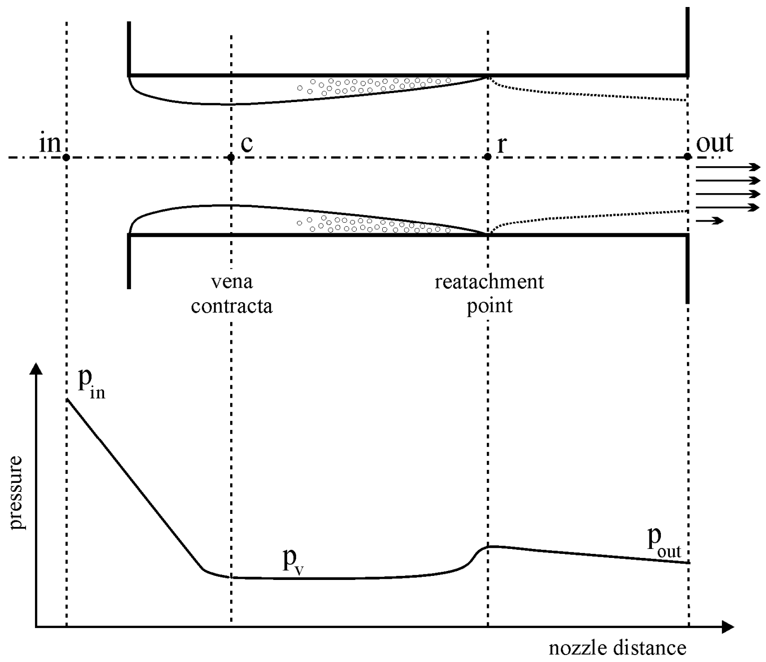


Figure 180: Onset of Cavitation in Nozzle Hole

(c) **Supercavitation**

For this regime along the walls a two-phase flow with very low density is assumed. Thus there are no losses except for the choking mechanism at point c (refer to Figure 181). To recover the pressure at the exit, the flow has to spread and slow down again.

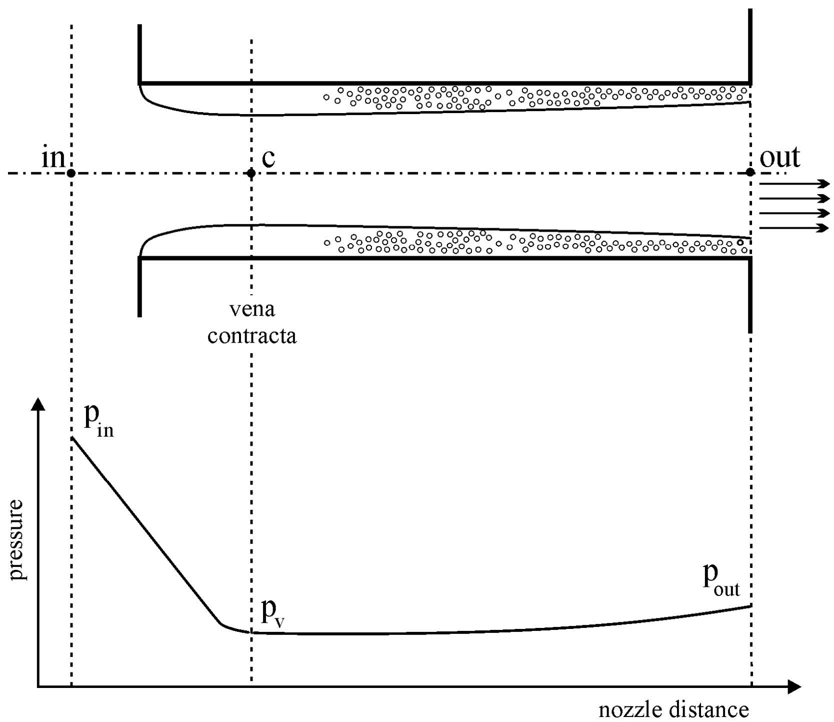


Figure 181: Supercavitation in Nozzle Hole

(d) **Calculation of flow-discharge coefficient**

<table>
<tr><td>Flow velocity:</td><td>$v_{inl} = \dfrac{Q_{hole}}{A_{inl}}\,;\, v_{outl} = \dfrac{Q_{hole}}{A_{outl}}$</td></tr>
<tr><td>Reynolds number:

Average Reynolds number:</td><td>$\mathrm{Re}_{inl} = \dfrac{v_{inl} D_{inl}}{\nu}\,;\ \mathrm{Re}_{outl} = \dfrac{v_{outl} D_{outl}}{\nu}$

$\mathrm{Re} = \dfrac{1}{2}\left(\mathrm{Re}_{inl} + \mathrm{Re}_{outl}\right)$</td></tr>
<tr><td>Wall friction:</td><td>$\lambda_{turb} = \dfrac{0.3164}{\mathrm{Re}^{0.25}}\,;\ \lambda_{lam} = \dfrac{64}{\mathrm{Re}}$
$\lambda = \max\left(\lambda_{turb}, \lambda_{lam}\right)$</td></tr>
</table>

<table>
<tr><th>Non Cavitating flow
($p_{vena} > p_{vap}$)</th><th>Cavitating flow
($p_{vena} \le p_{vap}$)</th></tr>
<tr><td rowspan="3">Correction of wall friction because of conical shape of spray hole:

$d_{ratio} = \dfrac{D_{inl}}{D_{outl}}$

$\lambda_{corr} = \dfrac{1}{4}\left(1 + d_{ratio} + d_{ratio}^2 + d_{ratio}^3\right)$

Flow discharge coefficient at non Cavitating flow:

$\boxed{C_D = \dfrac{1}{\sqrt{1. + K_{inlet} + \lambda_{corr}\lambda \dfrac{r_{inl}}{D_{inl}}}}}$

where K_{inlet} may be seen on Figure 182.</td><td>Flow contraction coefficient (for cylindrical spray hole):

$C_{C\_cyl} = \dfrac{1}{\sqrt{\left(\dfrac{1}{0.611}\right)^2 - 11.4\dfrac{r_{inl}}{D_{inl}}}}$

Correction of contraction coefficient (for conical spray hole):

$x_{Cc} = \dfrac{D_{inl}\left(D_{inl} - D_{outl}\right)}{4L_{hole}}$

$C_{C\_corr} = \dfrac{\left(D_{inl} - 2x_{Cc}\right)^2}{D_{inl}^2}$

Flow contraction coefficient (for conical spray hole):

$C_C = C_{C\_corr} C_{C\_cyl}$</td></tr>
<tr><td>Flow velocity at vena contracta:

$v_{vena} = \dfrac{v_{inl}}{C_c}$</td></tr>
<tr><td>Flow discharge coefficient at Cavitating flow:

$p_{inl} = p_{vapour} - \dfrac{1}{2}\rho v_{vena}^2$

$\boxed{C_D = \dfrac{v_{inl}}{\sqrt{\dfrac{2\left(p_{inl} - p_{outl}\right)}{\rho}}}}$</td></tr>
</table>

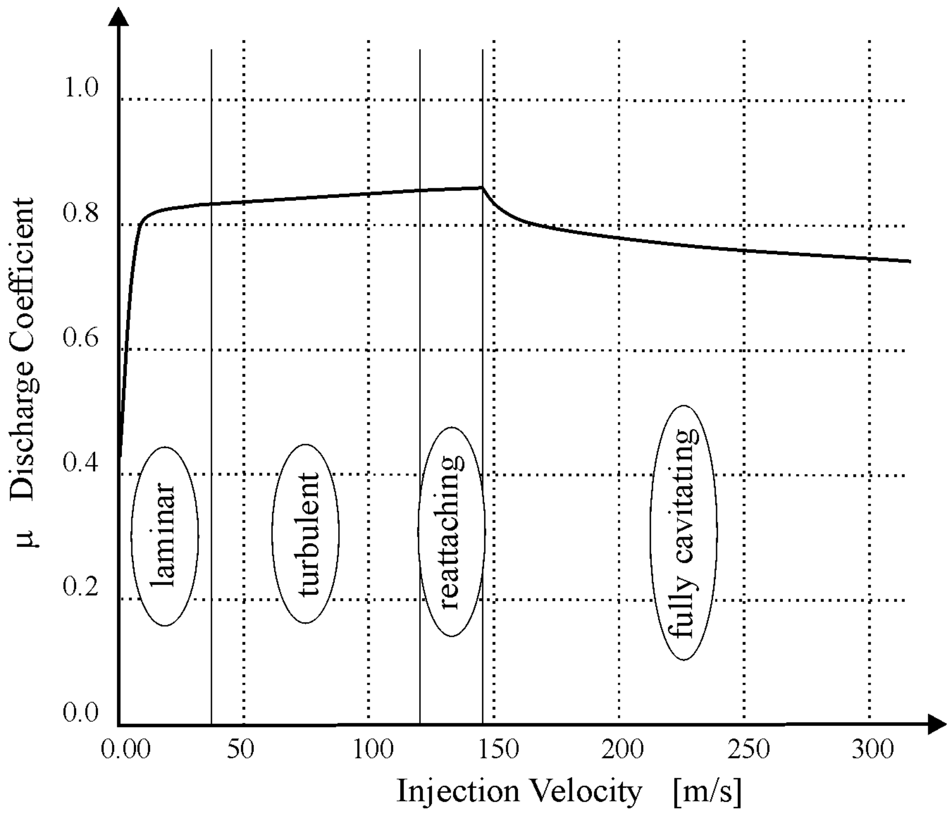


Figure 182: Flow Discharge Coefficient vs. Injection Velocity

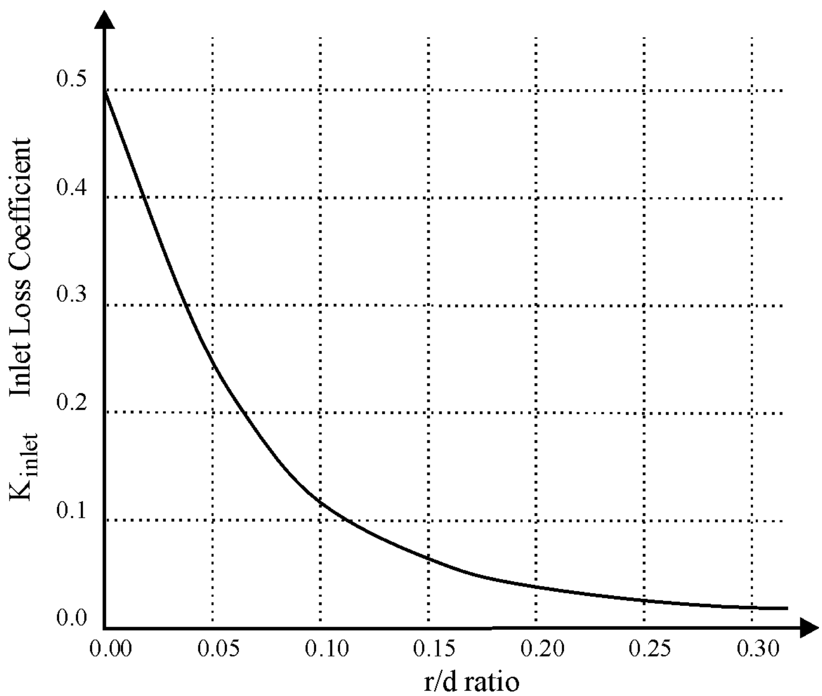


Figure 183: Turbulent Flow through Nozzle Hole

20.3.6. Model of Gas Flow through Nozzle Holes

The gas flow through **SAC Nozzle (Extended model)** can be simulated only in combination with **Two-phase Volume** and **Gas Boundary** elements as shown in Figure 190 Furthermore, in **Two-phase Volume** the **Fluid-Gas Mixture** option must be active. In this case BOOST Hydsim calculates the backward flow of gas through the nozzle holes if the pressure in **Gas Boundary** becomes higher than the pressure in the **Two-phase Volume**. **SAC Nozzle (Extended model)** uses gas properties from the **Cylinder Charge Properties** dialog box.

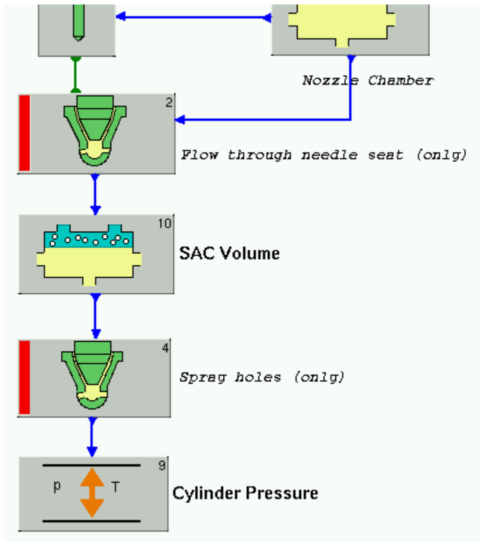


Figure 184: Gas Flow Model through Extended SAC Nozzle Orifice

The flow model through the nozzle holes is based on the energy equation, continuity equation and the equation for the isentropic change of state:

$$\dot{m} = \mu A_{holes} p_{cyl} \psi \sqrt{\frac{2}{R_g T_{cyl}}}, \qquad (400)$$

where $\dot{m}$ is the gas mass rate, μ is the flow discharge coefficient of the hole, p_{cyl} and T_{cyl} are pressure and temperature in Gas boundary (typically cylinder chamber), A_{holes} is the total cross-sectional area of nozzle spray holes and ψ is the function dependent on the flow type. Here only the subsonic flow is considered. For this, function ψ is given by the following equation:

$$\psi = \sqrt{\frac{\kappa}{\kappa - 1}\left[\left(\frac{p_{sac}}{p_{cyl}}\right)^{\frac{2}{\kappa}} - \left(\frac{p_{sac}}{p_{cyl}}\right)^{\frac{\kappa+1}{\kappa}}\right]}, \qquad (401)$$

where p_{sac} is the pressure in nozzle sac volume and κ is the specific heat ratio.

20.3.6.1. Multi-section Contour Geometry and Calculation of Minimum Seat Flow Areas

Needle tip/seat model with multi-section contour geometry calculates the minimum geometric and effective flow areas between the needle tip and nozzle body contours at any needle lift.

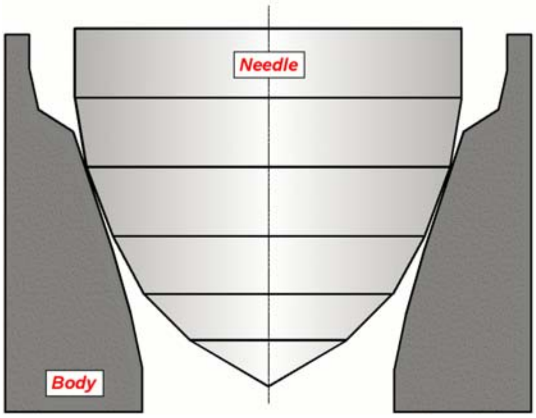


Figure 185: Nozzle Body with Needle

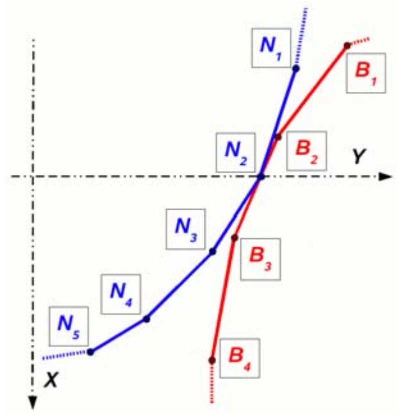
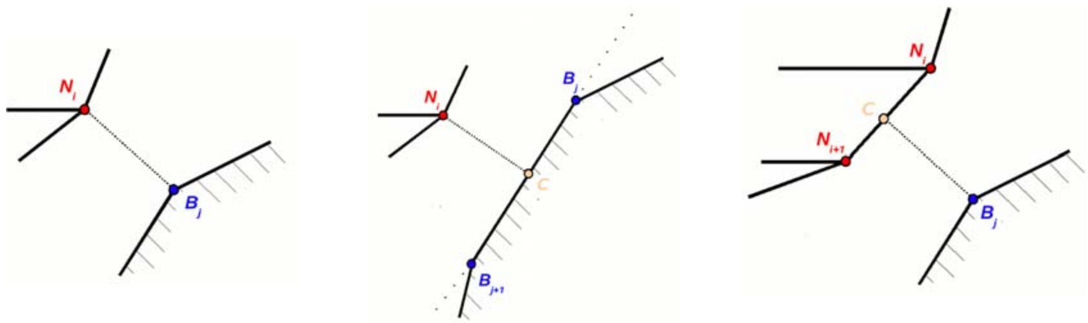


Figure 186: Initial Position of Needle Tip and Nozzle Body Contours

Algorithm consists of four major steps (loops) illustrated in Figure 187. First step is the loop over the edge points of needle tip and nozzle body contours searching for the minimum distance and area between them. Second step involves the loop over the edge points of the needle tip contour searching for the minimum distance and area between the needle edge and opposite nozzle body contour (truncated cone). Third step is the loop over the edge points of nozzle body searching for the minimum distance and area between the nozzle body edge and opposite needle tip contour (truncated cone). Finally, fourth step compares minimum geometric and effective areas from above loops with geometric and effective flow areas of nozzle sac.



1. Loop over edge points of needle tip and nozzle body contours

2. Loop over edge points of needle tip contour and nozzle body section

3. Loop over edge points of nozzle body contour and needle tip section

Figure 187: Major Steps of Minimum Flow Area Search Algorithm

20.3.6.2. BOOST Hydsim - FIRE Link at Needle Seat

The coupling between BOOST Hydsim and FIRE can be defined at/above the needle seat of the injection nozzle as shown in Figure 189.

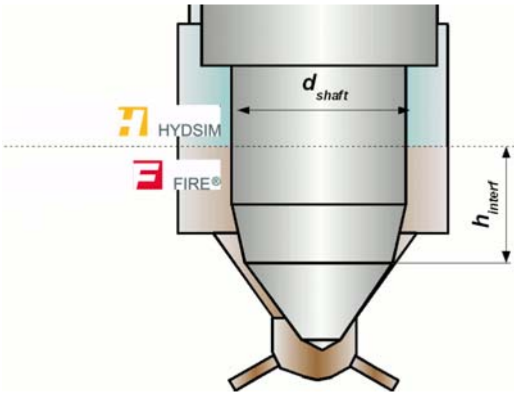


Figure 188: Interface between Fire and BOOST Hydsim

The basic co-simulation procedure at the BOOST Hydsim-FIRE interface is:

- FIRE transfers to BOOST Hydsim the resultant force $\boldsymbol{F} = (\boldsymbol{F_{Nx}}, \boldsymbol{F_{Ny}}, \boldsymbol{F_{Nz}})$ acting onto the needle tip which results from the three-dimensional pressure distribution around the tip up to a defined height $\boldsymbol{h_{interf}}$ ($\geq \mathbf{0}$) from the needle seat.
- Additionally, FIRE transmits to BOOST Hydsim the mass flow rate through needle seat and spray holes $\boldsymbol{q_{seat}}$, $\boldsymbol{q_{holes}}$, pressure and temperature in sac volume $\boldsymbol{p_{sac}}$, $\boldsymbol{T_{sac}}$ and pressure and temperature at the interface border $\boldsymbol{p_{topV}}$, $\boldsymbol{T_{topV}}$ defined by $\boldsymbol{h_{interf}}$.
- BOOST Hydsim computes the rigid-body needle movement for a coupling time step and tranfers to FIRE the displacement vector $\boldsymbol{d} = (\boldsymbol{dx}, \boldsymbol{dy}, \boldsymbol{dz})$.
- Additionally, BOOST Hydsim transmits to FIRE pressure and temperature in nozzle chamber $\boldsymbol{p_{nozV}}$, $\boldsymbol{T_{nozV}}$ (closest to 1D/3D interface border).
- BOOST Hydsim controls the co-simulation procedure by defining and sending to FIRE the data exchange start/end instants and time/angle step.

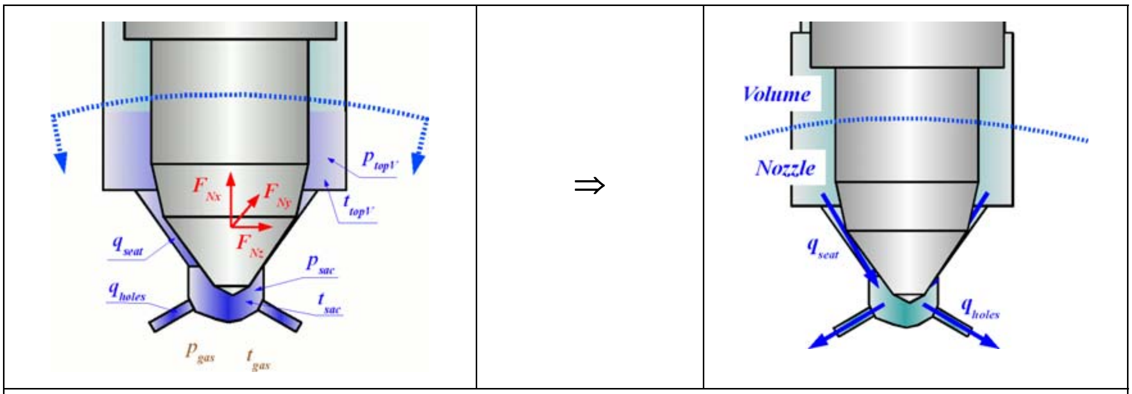
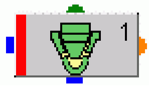


Figure 189: Flow through needle seat and nozzle holes calculated by Fire

20.4. Extended VCO Nozzle Orifice

Element Name:	**Extended VCO Nozzle Orifice**
Element Icon:	
Definition:	This element serves to define a Valve-covered Nozzle Orifice (extended model with cavitation).
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connection have to be defined. 

☞

Note: **VCO Nozzle Orifice** has to be connected by special connection to a **Needle** element. Via this connection **Nozzle Orifice** receives information about the actual **Needle** lift, and **Needle** element takes information about the pressure at needle seat area etc.

Note: Hydraulic connections to **VCO Nozzle Orifice** are irreversible: input connection implies a connection to nozzle volume (high pressure volume under needle guide) and output connection is the connection to the spray volume (e.g. combustion chamber, usually modeled as **Boundary** condition).

Note: For **Extended VCO Nozzle Orifice** connected to a **Fire Link** element, the flow through the nozzle and pressure distribution along needle tip is calculated by the FIRE solver (3D multi-phase flow simulation).

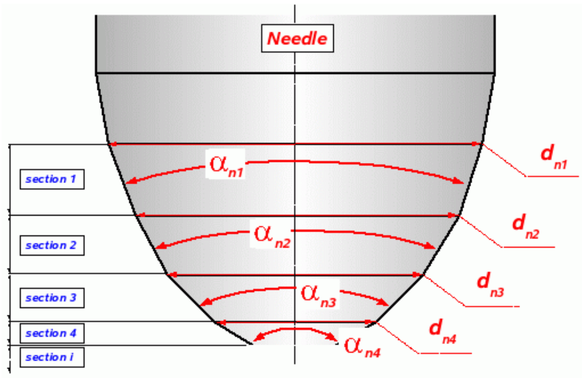
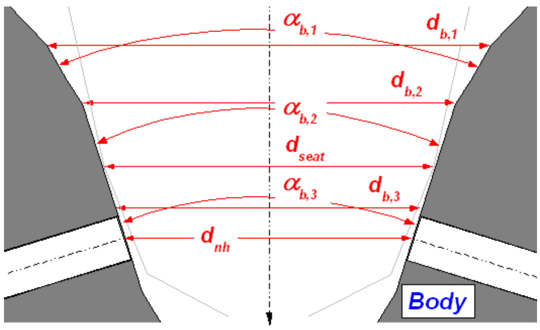
The cross-section of **VCO Nozzle Orifice** with spray holes and needle seat is shown in Figure 176 (refer to Section 20.2).

20.4.1. Input Parameters

Description of input data for the **Extended VCO Nozzle Orifice**:

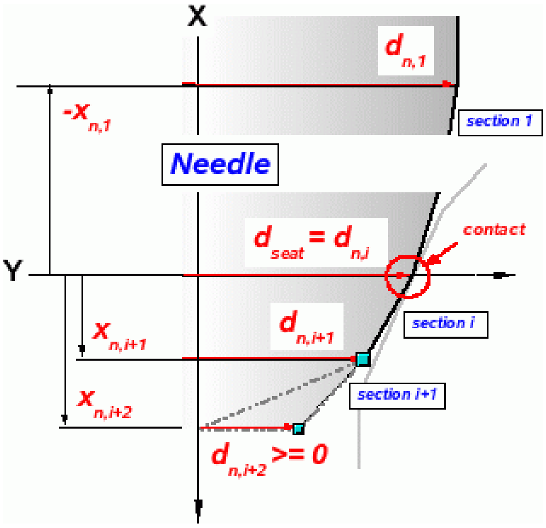
Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Spray Calculation	Press this bar to choose formulae for spray calculation (refer to Section 20.1.4). **Note:** If **Export to FIRE** is active, Spray cone angle calculation has to be activated, too.

FLOW THROUGH SPRAY HOLES	
Number of spray holes	Specify the number of spray holes of the nozzle N_{holes}.
Hole outlet diameter	Specify the outlet diameter of one spray hole d_{hole} (refer to Figure 176).
Spray hole length	Specify length of one spray hole l_{hole}. (refer to Figure 176). **Note:** Spray hole length is automatically enabled for the Spray cone angle calculation by **Sitkei** formula, Spray penetration calculation by **Varde, Popa** formula or in case of active „Program-calculated flow discharge coefficient“ option.
User-defined flow discharge coefficient	Click to activate user-defined flow discharge coefficient.
Hole discharge coefficient (no cavitation)	Specify flow discharge coefficient of nozzle spray holes at hydraulic flow.
Hole discharge coefficient at cavitation	Specify flow discharge coefficient of nozzle spray holes at cavitating flow μ_{cav} (spray hole exit completely filled with the liquid vapor). The cavitation model is described in Chapter 17.2.6 and Egartner[35].
Discharge coefficient at hole inlet	Specify flow discharge coefficient at spray hole inlet μ_{inlet}.
Program-calculated flow discharge coefficient	Click to activate program-calculated flow discharge coefficient (refer to Section 20.3.5.2).
k-factor	Specify *k-factor* for conical holes: $$k - factor = \frac{D_{inlet} - d_{outlet}}{10\mu m}$$ It may be defined only for „Program-calculated flow discharge coefficient“ case.
Hole inlet radius	Specify inlet radius of nozzle spray hole.
Vapor pressure	Specify fluid vapor pressure in the spray hole.
FLOW THROUGH NEEDLE SEAT	
Needle Tip/Seat Geometry	
Standard	
Angle of needle seat	Specify angle of needle seat α_{seat} (refer to Figure 176).
Nozzle diameter at spray holes	Specify nozzle diameter at nozzle holes location d_{nsh} (refer to Figure 176).
Multi-section Contour	(Model is described in Chapter 17.3.7)
Input type: Section angle	

<table>
<tr><td>Needle Tip</td><td>Needle tip contour table:
Specify base diameter d_{ni} of truncated cone (frustum) in the 1st column in descending order
Specify cone angle α_{ni} of tip frustum in the 2nd column in ascending order
Specify discharge coefficient μ_{ni} at the flow area near respective frustum base (tip contour edge) (*1)

</td></tr>
<tr><td>Diameter of needle tip end</td><td>Specify diameter of needle tip end d_{end}.</td></tr>
<tr><td>Nozzle Body</td><td>Nozzle body contour table:
Specify base diameter d_{bi} of truncated cone (frustum) in the 1st column in descending order
Specify cone angle α_{bi} of body frustum in the 2nd column in descending order

</td></tr>
<tr><td>Diameter at spray holes</td><td>Specify diameter at spray holes.
Note: diameter at spray holes implies the bottom diameter of the nozzle body contour defined in above table.</td></tr>
<tr><td>Flow discharge at spray hole inlet (*1)</td><td>Specify flow discharge coefficient at spray hole inlet.</td></tr>
<tr><td colspan="2"><u>Multi-section Contour</u>
Input type: Base coordinate</td></tr>
</table>

<table>
<tr><td>Needle Tip</td><td>
Needle tip contour table:

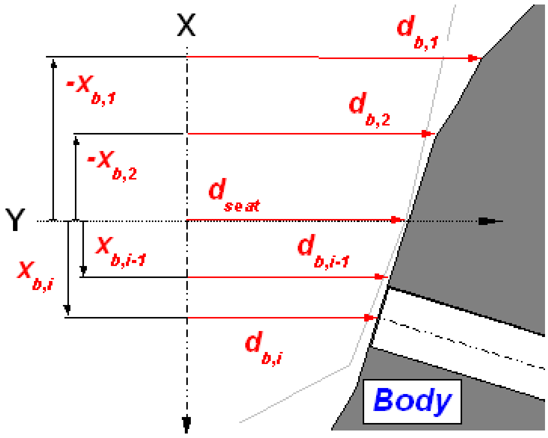
Specify diameter d_{ni} of frustum (truncated cone) base in the 1st column (arbitrary geometry)
Specify x-coordinate x_{ni} of frustum base in the 2nd column in ascending order
Specify discharge coefficient μ_{ni} at the flow area near respective frustum base (tip contour edge)



Notes (input requirements):

Coordinate system origin is the intersection point between the vertical needle symmetry axis x and seat plane (horizontal axis y)
For a sharp needle tip (usual case), base diameter in last row has to bet set to zero
Sign definition rule of base x-coordinate in 2nd column: for base above seat x-coordinate is negative (–x), at needle seat x-coordinate is zero (x=0, this input row must exist) and for base below seat x-coordinate is positive (+x)
Flow coefficient of any effective area inside needle tip section is calculated by the linear interpolation of the flow coefficients between the neighboring table rows (contour edges). This coefficient applies for turbulent flow calculation only
Flow coefficient in last table row has to be the inlet discharge coefficient of nozzle sac

</td></tr>
</table>

Nozzle Body	Nozzle body contour table: • Specify base diameter d_{bi} of truncated cone (frustum) in the 1st column in non-ascending order • Specify x-coordinate x_{bi} of frustum base in the 2nd column in ascending order  **Notes (input requirements):** 1. Coordinate system origin is the intersection point between the vertical needle symmetry axis x and seat plane (horizontal axis y) 2. Nozzle body contour has to be defined till spray hole's inlets 3. Base diameter in last table row has to be diameter at spray holes (bottom diameter of body contour) 4. Sign definition rule of base x-coordinate in 2nd column: for base above seat x-coordinate is negative ($-x$) and for base below seat x-coordinate is positive ($+x$) 5. Nozzle body contour cannot contain an edge at needle seat (i.e. input x=0 is not allowed)
Loss coefficient at needle seat (*1)	
Constant for laminar flow	Specify loss constant in the numerator C_{lam} of the equation below (also refer to Equation 389 in Section 20.3.5).
Exponent of Reynolds number	Specify the exponent γ for Reynolds number (Re). Definition of loss coefficient for laminar flow ξ_{lam}: $$\xi_{lam} = \frac{C_{lam}}{\mathrm{Re}^{\gamma}}$$
Turbulent flow coefficient	Press this bar to specify turbulent flow coefficient ξ_{turb}. It may be constant or function of needle lift. **Note:** Turbulent flow coefficient is inactive for "Multi-section Contour" option. In this case turbulent flow coefficients are defined in 3rd column of Needle tip contour table.
Constant (*1)	
Turbulent flow constant at needle seat	Specify loss coefficient for turbulent flow (if appropriate).

Needle Lift-dependent	• specify lift of needle in ascending order in the first column. • specify turbulent flow coefficient at opening in the second column. • specify turbulent flow coefficient at closing in the third column. For needle lifts higher than the highest value specified, the last ξ_{turb} value from the table is used. For lifts lower than the lowest value specified, the first ξ_{turb} value from the table is taken (kept constant as long as needle lift is out of table range).
(*1) **Note:** For **VCO Nozzle Orifice** connected to a Fire Link element, some parameters are disabled (grayed out) because they are not necessary for a coupled BOOST Hydsim-FIRE calculation.	

20.4.2. Initial Conditions

Initial conditions cannot be specified for **Extended VCO Nozzle Orifice**.

20.4.3. Modifiable Parameters

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name has to be pressed. When modification is activated, Modification Table has to be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

hole discharge coefficient (no cavitation)	SI Unit:	---
hole discharge coefficient (at cavitation)	SI Unit:	---
constant for laminar flow at needle seat	SI Unit:	---
turbulent flow constant at needle seat	SI Unit:	---
flow discharge coef. at hole inlet	SI Unit:	---

20.4.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

volumetric flow rate through nozzle orifice	Volumetric flow rate through nozzle orifice is calculated from Equation 386.
volumetric flow rate through nozzle orifice	Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow rate through nozzle orifice	Cumulative rate is injection quantity per calculation range.

pressure at spray hole inlet	Pressure in the area before nozzle spray holes is calculated according to Equation 404. or 405.
effective flow area at needle seat	Cross-sectional flow area at needle seat is calculated according to geometrical characteristics of needle seat. It is the minimum cross sectional flow area between needle and seat calculated from Equation 387.
cavitation switch	Possible values: 0 – flow (needle closed); 1 – hydraulic flow; 2 – cavitating flow
cavitation factor in nozzle orifice	Cavitation factor is calculated from the relationships in Equations 391 and 392 located in Section 20.3.5.
effective flow area at 'vena contracta'	for turbulent flow: $$A_{vena} = \frac{A_{hole\_i}}{\sqrt{\frac{1}{\mu_{inlet}^2} + \frac{1}{\mu_{hyd}^2}}}$$ for cavitating flow: $$A_{vena} = \mu_{cav} A_{hole\_i}$$
effective flow area at spray hole exit	for turbulent flow: $$A_{eff} = A_{hole\_i}$$ for cavitating flow: $$\mathrm{v}_{mean} = \frac{Q}{A_{hole\_i}}$$ $$\mathrm{v}_{vena} = \sqrt{\frac{2p_{in}}{\rho}}$$ $$\mathrm{v}_{eff} = \mathrm{v}_{vena} - \frac{p_{out}}{\rho \mathrm{v}_{mean}}$$
mass flow rate through nozzle orifice	Mass flow rate through nozzle orifice is calculated from the equation: $$m = Q\rho_D \, .$$
mass flow rate through nozzle orifice	Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative mass flow through nozzle orifice	Cumulative mass rate is the injected mass per calculation range.
spray ...	Refer to Section 20.1.4

20.4.5. Flow Equations

The flow model through **Extended VCO Nozzle Orifice** is shown in Figure 190. In principle, it is similar to the flow model through **Extended SAC Nozzle Orifice** shown in Figure 178. The flow through needle seat and nozzle spray holes is represented by two in series switched one-dimensional orifices.

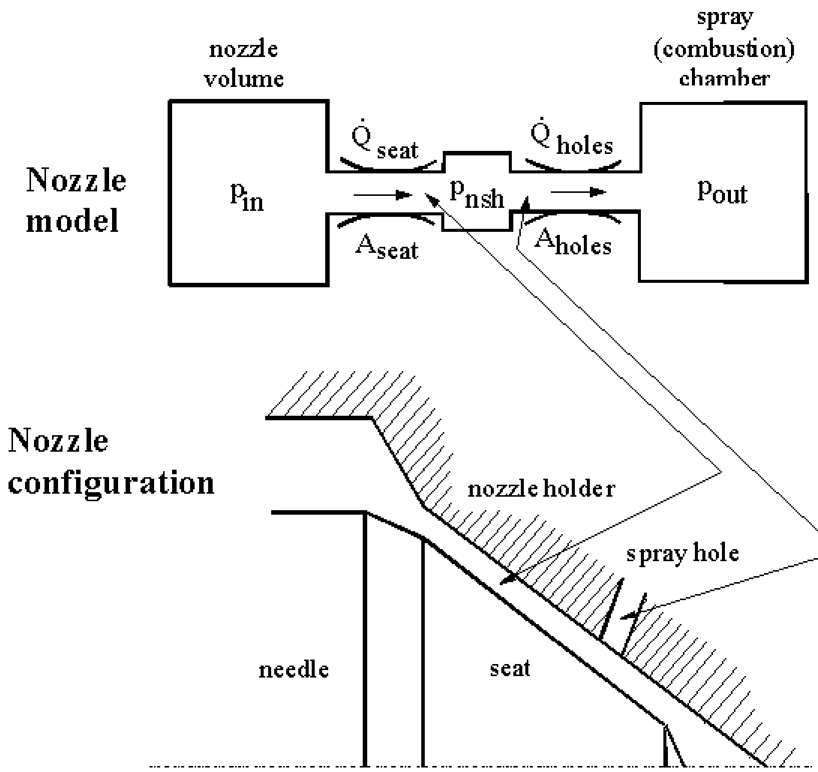


Figure 190: Flow Model through Extended VCO Nozzle Orifice

Volumetric flow rate (injection rate) through **Extended VCO Nozzle Orifice** is calculated identically as for the **Extended SAC Nozzle Orifice** (refer to Equation 386 in Section 20.3.5). However, narrowest cross-sectional flow area at needle seat A_{seat} is estimated as the envelope area of the conical gap with the base diameter equal to the average of the needle seat diameter and nozzle diameter at the location of spray holes. Thus, A_{seat} is obtained from the following equation:

$$A_{seat} = \frac{1}{2}\pi x_{lift} \sin\frac{\alpha_{seat}}{2}\left(d_{nsh} + d_{seat}\right), \tag{402}$$

where x_{lift} is the needle lift, α_{seat} is the angle of needle seat and d_{nsh} is the nozzle diameter at the location of spray holes. Needle lift of a **Standard** and **Two-spring Injector Nozzle** is calculated from the differential equation of motion in Chapter 19.

Furthermore, hydraulic diameter of the flow pattern (stream tube) through needle seat is calculated differently:

$$D_h = 2x_{lift} \sin\frac{\alpha_{seat}}{2}. \tag{403}$$

All other equations and statements from the Section 20.3.5 are also valid for the **Extended VCO Nozzle Orifice.**

Note that pressure in the nozzle sac of **Extended SAC Nozzle Orifice** is equivalent to the pressure under the needle seat p_{seat} (for the **up-to-date Needle** model) or pressure under the area of the spray holes p_{nsh} (for the **obsolete Needle** model) of **Extended VCO Nozzle Orifice**. This pressure depends on the cavitation condition. For hydraulic flow, satisfying Equation 391, it is calculated from the equation:

$$p_{seat/nsh} = p_{out} + \left(\frac{\xi_{inlet}}{A_{holes}^2} + \frac{\xi_{holes}}{A_{holes}^2} \right) \dot{Q}^2 \frac{\rho}{2}. \tag{404}$$

For cavitating flow, satisfying Equation 392, pressure under the needle seat (nozzle spray holes) is given by:

$$p_{seat/nsh} = p_{in} - \frac{\xi_{seat}}{A_{seat}^2} \dot{Q}^2 \frac{\rho}{2}. \tag{405}$$

For more information on the cavitation phenomenon in **Extended VCO Nozzle Orifice**, refer to Chapter 17.

20.4.5.1. Multi-section Contour Geometry and Calculation of Minimum Seat Flow Area

Needle tip/seat model with multi-section contour geometry calculates the minimum geometric and effective flow areas between the needle tip and nozzle body contours at any needle lift.

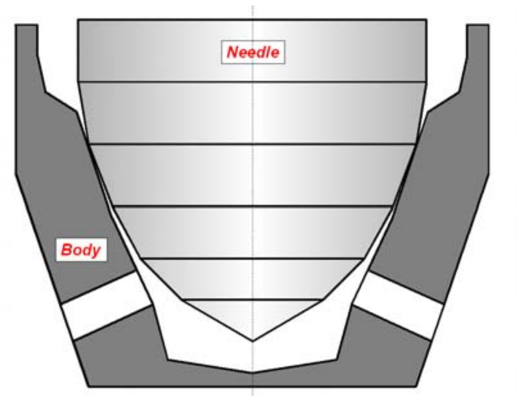


Figure 191:
VCO Nozzle Body with Needle

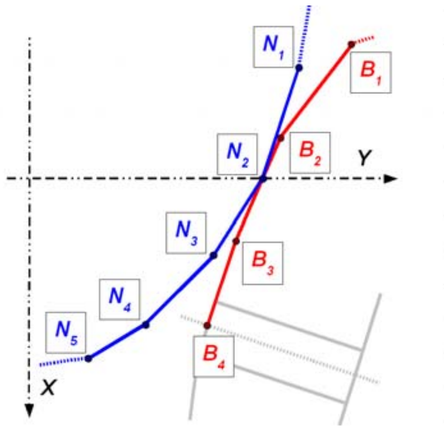
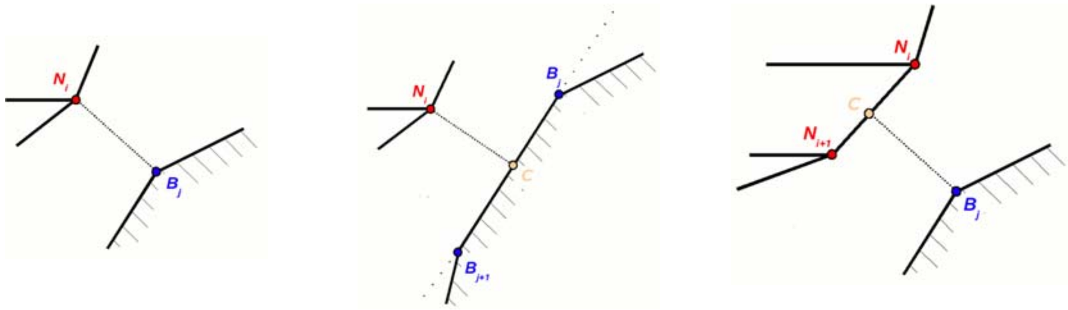


Figure 192:
Initial Position of Needle Tip and Nozzle Body Contours

Algorithm consists of the four major steps (loops) illustrated in Figure 193. The first step is the loop over the edge points of needle tip and nozzle body contours searching for the minimum distance and area between them. The second step involves the loop over the edge points of the needle tip contour searching for the minimum distance and area between the needle edge and opposite nozzle body contour (truncated cone). The third step is the loop over the edge points of nozzle body searching for the minimum distance and area between the nozzle body edge and opposite needle tip contour (truncated cone). Finally, the fourth step compares minimum geometric and effective areas from above loops with the geometric and effective inflow areas into nozzle holes.



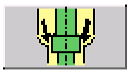
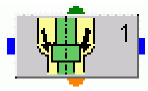
1. Loop over edge points of needle tip and nozzle body contours

2. Loop over edge points of needle tip contour and nozzle body section

3. Loop over edge points of nozzle body contour and needle tip section

Figure 193: Major Steps of Minimum Flow Area Search Algorithm

20.5. RSN Throttle

Element Name:	**RSN Throttle**	
Element Icon:		
Definition:	This element serves to define the properties of needle collar throttle of rate-shaping nozzle (RSN).	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 1 wire pins: 1 Only one input and one output hydraulic connection have to be defined.	

Note: Currently the Variable-step solver does not support this element.

Note: Hydraulic connections to **RSN Throttle** are irreversible: input connection implies a connection to nozzle volume (high pressure volume under needle guide) and output connection is the connection to the spray volume (e.g. combustion chamber, usually modeled as **Boundary** condition).

The RSN Throttle geometry is shown in Figure 194.

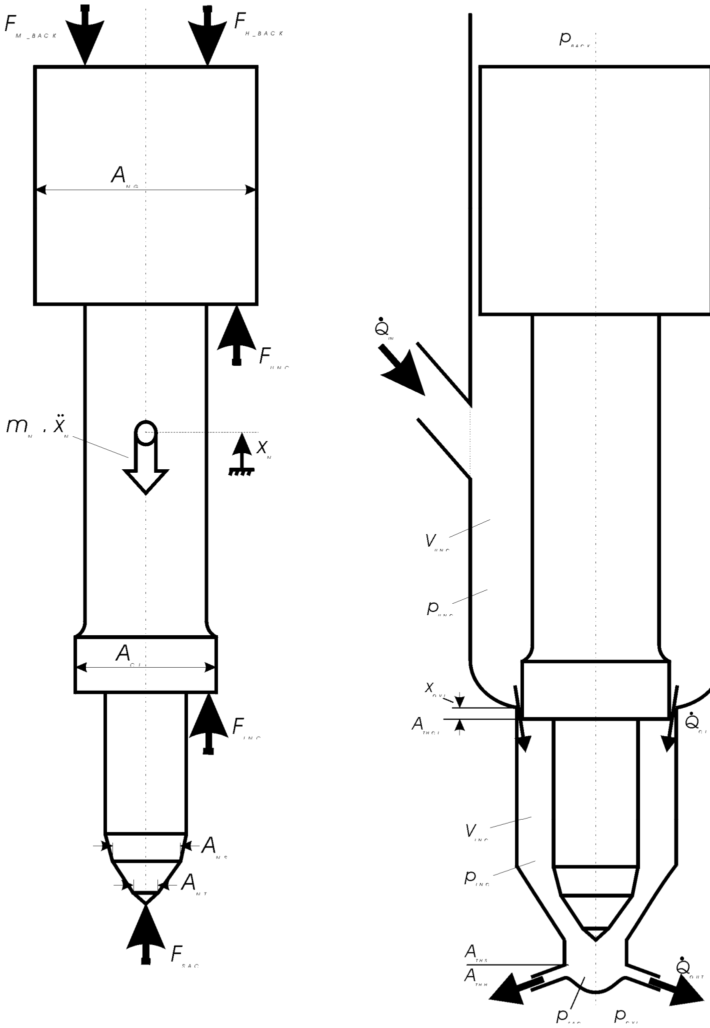


Figure 194: RSN Throttle Geometry

20.5.1. Input Parameters

Description of input data for the **RSN Throttle**:

Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section 8.1.4).
Diameter of needle collar	Specify diameter of needle collar d_{cl} (refer to Figure 194)
Initial collar overlap	Specify initial overlap of needle collar x_{ovl} (refer to Figure 194)

PROGRAM-CALCULATED FLOW AREA (standard geometry)	If this button is active, flow area is calculated from the geometric characteristics (refer to Section 20.5.6)
Diametral gap at overlap	Specify gap width between needle collar and nozzle wall at overlap y_{cl} (refer to Figure 194)
USER-DEFINED FLOW AREA (special collar geometry)	If this button is active, flow area is obtained from the table of the flow area vs. needle lift.
Effective Flow Area (my*A)	
Factor for 2nd column	This button activates the multiplication factor for 2nd column of table.
	Values in second column will be multiplied with the scale factor as soon as **Run / Run Sets / Restart** is pressed in **Simulation** menu. It is convenient to use 'Factor for 2nd column' as discharge coefficient (if it is constant and seat closing/ opening areas are available in absolute values not premultiplied by μ).
FLOW AREA TABLE	Specify needle lift in ascending order in the first column. Specify effective cross-sectional flow area (or directly area if μ is used as a 'Factor for 2nd column') in the second column. For needle lifts higher than the highest value specified, the highest μA or $\dot{Q}$ value from the table is used. For lifts lower than the lowest value specified, the lowest μA or $\dot{Q}$ value from the table is taken (kept constant as long as needle lift is out of table range).
Flow Resistance	Press this bar to specify flow resistance coefficient (refer to Section 20.5.4)

20.5.2. Initial Conditions

Initial conditions cannot be specified for **RSN Throttle**.

20.5.3. Modify Parameter

Modifiable Parameters cannot be specified for **RSN Throttle**.

20.5.4. Flow Resistance

This dialog serves to define parameters for flow resistance coefficient calculation.

Description of input data:

FLOW RESISTANCE COEFFICIENT	
Constant for laminar flow C_lam	Specify loss constant in the numerator C_{lam} of the equation below (also refer to Equation 389 in Section 20.3.5).

Exponent of Reynolds number n	Specify the power factor γ for Reynolds number (Re). Definition of loss coefficient for laminar flow ξ_{lam}. $$\xi_{lam} = \frac{C_{lam}}{\mathrm{Re}^{\gamma}}$$
Turbulent flow constant C_turb	Specify loss coefficient for turbulent flow (if appropriate).

20.5.5. Output Parameters

Path: **Element**| **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

volumetric flow rate through valve	Volumetric flow rate through nozzle orifice is calculated from Equation 406.
volumetric flow rate through valve	Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow rate through valve	Cumulative rate is injection quantity per calculation interval.
effective cross-section valve flow area	Corresponds to μA.
hydraulic force on RSN needle collar	Hydraulic force acting on the needle collar is calculated as follows. **Obsolete Needle** model: $\Rightarrow$ for $x_{lift} \le 0$ $$F_{cl} = p_{inp}\left(A_{cl} - A_{seat}\right),$$ $\Rightarrow$ for $x_{lift} > 0$ $$F_{cl} = p_{inp}\left(A_{cl} - A_{tip}\right);$$ Up-to-date Needle model: $$F_{cl} = p_{inp}\left(A_{cl} - A_{seat}\right),$$ where A_{cl} is the needle collar area ($A_{cl} = \frac{1}{4} d_{cl}^2 \pi$), A_{seat} is the needle seat area and A_{tip} is the needle tip area. **Note: RSN Nozzle** and **Needle** elements has to be connected via special connection.

20.5.6. Flow Equations

Volumetric flow rate through **RSN Throttle** is calculated from the formula:

$$Q = \sqrt{\frac{1}{\xi_{cl}}} A_{cl} \sqrt{\frac{2}{\rho_{av}} \left(p_{unc} - p_{lnc} \right)}, \tag{406}$$

where ξ_{cl} is the flow resistance coefficient, A_{cl} is the open cross-sectional flow area of the needle collar throttle, ρ_{av} is the average fluid density and p_{unc} and p_{lnc} are the pressures in the upper an lower nozzle chambers, respectively. Cross-sectional flow area of the collar throttle is given by:

$$\forall x_{lift} \le x_{ovl} :$$

$$A_{cl} = \frac{\pi}{4} \left(2 d_{cl} y_{cl} + y_{cl}^2 \right), \tag{407}$$

$$\forall x_{lift} > x_{ovl} :$$

$$A_{cl} = \pi \left(d_{cl} + \frac{y_{cl}}{2} \right) \sqrt{\left(x_{lift} - x_{ovl} \right)^2 + \left(\frac{y_{cl}}{2} \right)^2}, \tag{408}$$

where x_{ovl} is the initial overlap and y_{cl} is the diametral gap at the needle collar.

20.6. Non-circular Needle Guide Throttle

<table>
<tr><td>Element Name:</td><td colspan="2">Non-circular Guide Throttle</td></tr>
<tr><td>Element Icon:</td><td colspan="2"></td></tr>
<tr><td>Definition:</td><td colspan="2">This element serves to define the flow properties through non-circular needle guide with segments (for the fuel inflow).</td></tr>
<tr><td>Connecting pins:</td><td>standard pins: 2 (hydraulic)
special pins: 1
wire pins: 1
Only one input and one output hydraulic connection have to be defined.</td><td></td></tr>
</table>

☞ **Note:** Currently the Variable-step solver does not support this element.

The Non-circular Needle Guide Throttle is shown on Figure 195. Cross-section of needle guide cut is shown in Figure 199:

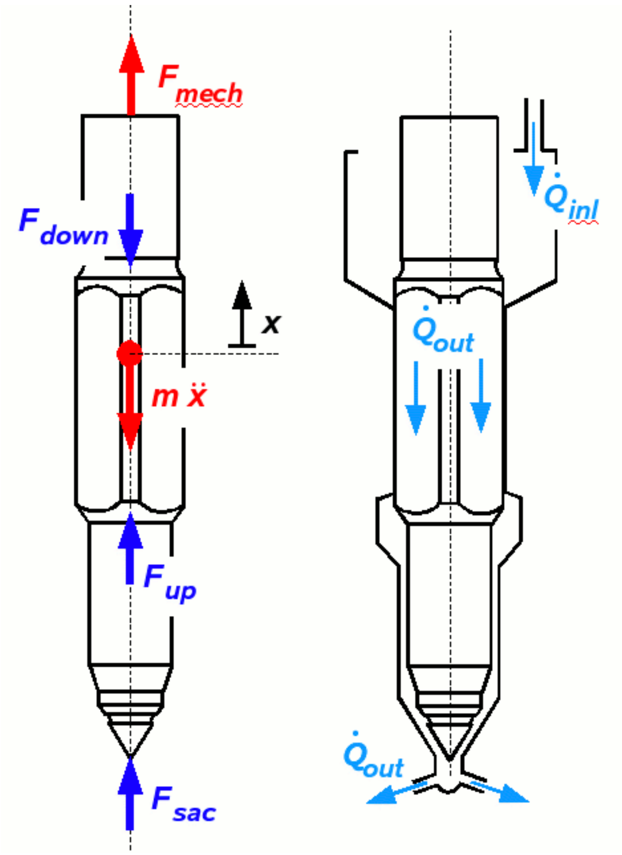


Figure 195: Flow Model of Non-circular Needle Guide Throttle

20.6.1. Input Parameters

Description of input data for the **Non-circular Guide Throttle**:

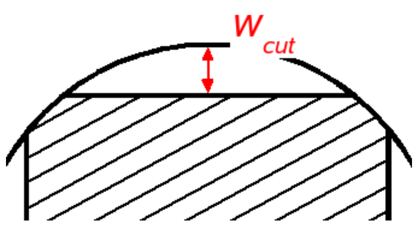
Element name	The name of the element is specified as default.
Fluid Properties	Press this bar to specify the local fluid properties (refer to Section **26.2.4.4**).
NEEDLE GUIDE CUT GEOMETRY	
<u>Segment</u>	
Segment width (from chord)	Specify segment width from chord w_{cut}.
Cut length (along guide axis)	Specify axial length of needle cut l_{cut}.
Number of guide cuts	Specify number of guide cuts N.
	

Figure 196: Simple type of needle guide cut (A-A)	
Double Segment	
Outer segment width (from chord)	Specify outer segment width from chord w_{cut}.
Inner segment curvature radius	Specify inner segment curvature radius r_{inn}.
Cut length (along guide axis)	Specify axial length of needle cut l_{cut}.
Number of guide cuts	Specify number of guide cuts N.

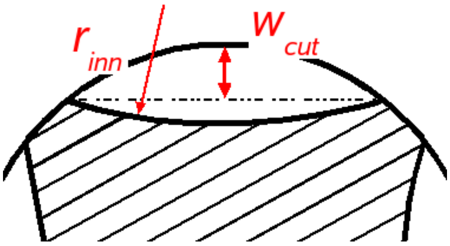


Figure 197: Double segment type of needle guide cut (A-A)

Other	
Cross-sectional area	Specify cross-sectional area for all segments A.
Number of guide cuts	Specify number of guide cuts N.

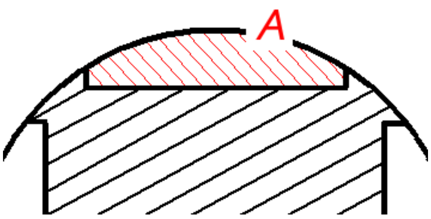


Figure 198: Other type of needle guide cut (A-A)

FLOW RESISTANCE	
Laminar flow coefficient	Press to activate the laminar flow calculation.
Proportional constant (C_lam)	Specify laminar flow coefficient C_{lam}.
Exponent of Reynolds number (n)	Specify exponent of Reynolds number n.
Turbulent flow coefficient (constant)	If this button is active, turbulent flow coefficient will be taken constant.
Resistance coefficient (C_turb)	Specify turbulent flow coefficient C_{turb}.

Turbulent flow coefficient (variable)	If this button is active, turbulent flow coefficient will be obtained from the table of the turbulent coef. vs. needle lift.
C_turb Table	Specify needle lift in ascending order in the first column. Specify turbulent flow coefficient in the second column. For needle lifts higher than the highest value specified, the highest C_{turb} value from the table is used. For lifts lower than the lowest value specified, the lowest C_{turb} value from the table is taken (kept constant as long as needle lift is out of table range).

20.6.2. Initial Conditions

Initial conditions cannot be specified for **Non-circular Guide Throttle**.

20.6.3. Modify Parameter

Modifiable Parameters cannot be specified for **Non-circular Guide Throttle**.

20.6.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the option button in front of output parameter has to be checked.

Description of output parameters:

volumetric flow rate through guide /time	Volumetric flow rate through guide is calculated from Equation 411.
volumetric flow rate through guide /angle	Output values will be 0, if the **Time** domain button is active in **Calculation Control** dialog box.
cumulative flow rate through guide	Cumulative rate is injection quantity per calculation interval.
Reynolds number	Reynolds number (*Re*) through guide is calculated from Equation 409.
flow resistance coefficient in throttle	Flow resistance coefficient (ξ_{cu}) in throttle is calculated for Equation 410.
pressure drop across guide	Pressure difference between p_{spr} and p_{nz}.

20.6.5. Flow Equations

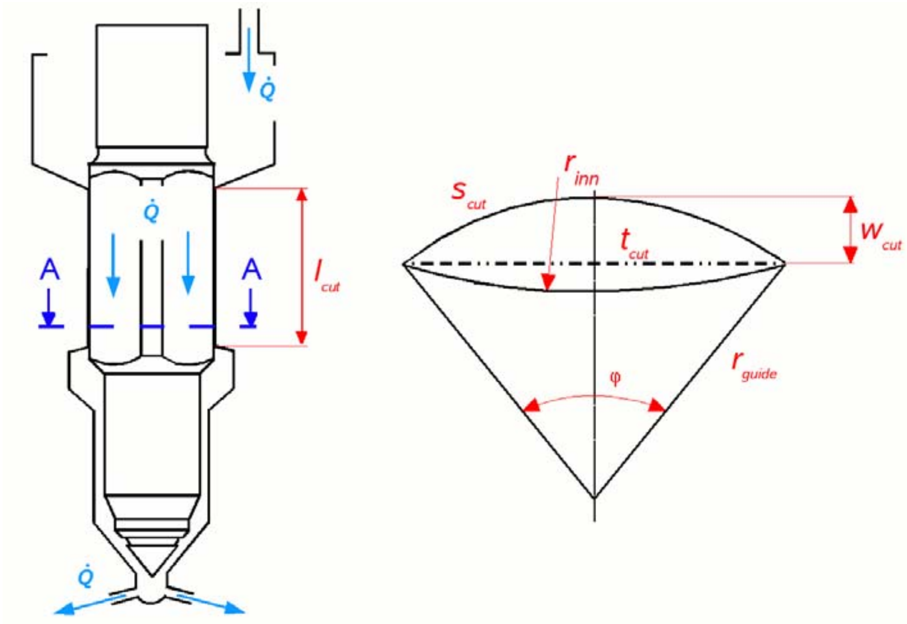


Figure 199: Needle Guide Cut Geometry

Cross-sectional area A_{cut} and hydraulic diameter D_h are calculated from the following formulae:

$$r_{guide} = \frac{1}{2} d_{guide}$$

$$\widehat{\varphi}_{out} = 2 \arccos\left(1 - \frac{w_{cut}}{r_{guide}}\right)$$

$$t_{cut} = 2 r_{guide} \sin \frac{\widehat{\varphi}_{out}}{2}$$

$$\widehat{\varphi}_{in} = 2 \arcsin \frac{t_{cut}}{2 r_{inn}}$$

$$s_{cut}^{in} = r_{inn} \widehat{\varphi}_{in}$$

$$s_{cut}^{out} = r_{guide} \widehat{\varphi}_{out}$$

$$A_{cut}^{in} = \frac{1}{2} r_{inn}^2 \left(\widehat{\varphi}_{in} - \sin \widehat{\varphi}_{in}\right)$$

$$A_{cut}^{out} = \frac{1}{2} r_{guide}^2 \left(\widehat{\varphi}_{out} - \sin \widehat{\varphi}_{out}\right)$$

Segment	Double Segment
$D'_h = \frac{4A_{cut}^{out}}{s_{cut}^{out} + t_{cut}}$	$D'_h = \frac{4\left(A_{cut}^{in} + A_{cut}^{out}\right)}{s_{cut}^{in} + s_{cut}^{out}}$
$A_{cut} = NA_{cut}^{out}$	$A_{cut} = N\left(A_{cut}^{in} + A_{cut}^{out}\right)$
$D_h = ND'_h$	$D_h = ND'_h$

Reynolds number for the calculation of laminar part of flow resistance coefficient is obtained from the formula:

$$\mathrm{Re} = \frac{QD_h}{\upsilon A_{cut}}, \qquad (409)$$

where ν is the kinematic viscosity of the fluid.

Flow resistance coefficient ξ_{cutt} in Equation 411 depends on the flow type (laminar or turbulent). It is calculated from the following condition

$$\xi_{cut} = \frac{C_{lam}}{\mathrm{Re}^n} + C_{turb}, \qquad (410)$$

For other types of needle guide geometry, the first term in flow resistance equation is set to zero and cross-sectional area is a product of the segment area and number of segments.

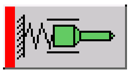
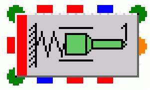
Volumetric flow rate through **Non-circular Guide Flow** is calculated from the equation:

$$Q = \sqrt{\frac{1}{\xi_{cut}}} A_{cut} \sqrt{\frac{2}{\rho_{av}}\left(p_{spr} - p_{nz}\right)}, \qquad (411)$$

where ξ_{cut} is the flow resistance coefficient, A_{cut} is the cross-sectional flow area of the needle cut, ρ_{av} is the average fluid density and p_{spr} and p_{nz} are the pressures in the spring and nozzle chambers, respectively.

21. NEEDLE

21.1. Standard Needle

Element Name:	**Standard Needle**
Element Icon:	
Definition:	This element defines the properties of a Standard Needle.
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 3 wire pins: 1 

Note: **Needle** element may have 2 hydraulic and up to 5 mechanical connections (in x direction). These connections are irreversible and have to be specified according to the following rules:

- **Needle** must have an **input** hydraulic connection from the **Nozzle Volume** (high pressure volume under needle guide). **Output** hydraulic connection to the outlet (spring) **Volume** or **Pressure** boundary is optional.
- **Needle** spring must be defined as an output mechanical connection to a **Mechanical** boundary (nozzle holder) or **Hydromechanical** boundary.
- In case of **Hydromechanical** boundary, mechanical connection is also a hydraulic connection, i.e. it transmits boundary pressure (if any) to **Needle**.

Note: **Standard Needle** model is based on the commonly accepted assumption that at needle opening the nozzle volume pressure is acting till the needle seat and sac pressure is acting under the needle seat. Flow throttling is always effective at needle seat. It is ultimately **recommended for common rail** systems.

Note: For creating a complete nozzle model, **Needle** must be connected to one of the **Nozzle Orifice** elements by a special connection.

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **Needle** may have only one hydraulic connection on each end.

Note: Mechanical connections can be defined only in local x-direction.

Note: Local x-axis of **Needle** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter Direction of Element motion.

☞

Note: For **Needle** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system too.

Note: Rigid **Needle** element can have 1 (translation in x-direction) or 3 degrees of freedom (translation in x, y and z directions). Currently 3-DOF **Needle** can be used only with **FIRE-Link (Nozzle)** element.

Note: With a **FIRE-Link (Nozzle)** element connected to **Needle**, both 1-DOF and 3-DOF **Needle** element can be used. As pressure distribution along needle seat is calculated by FIRE, the option "Distributed pressure at needle seat" is not applicable in this case.

The cross-section of a **SAC Nozzle** with a **Standard Needle** geometry is shown in Figure 200.

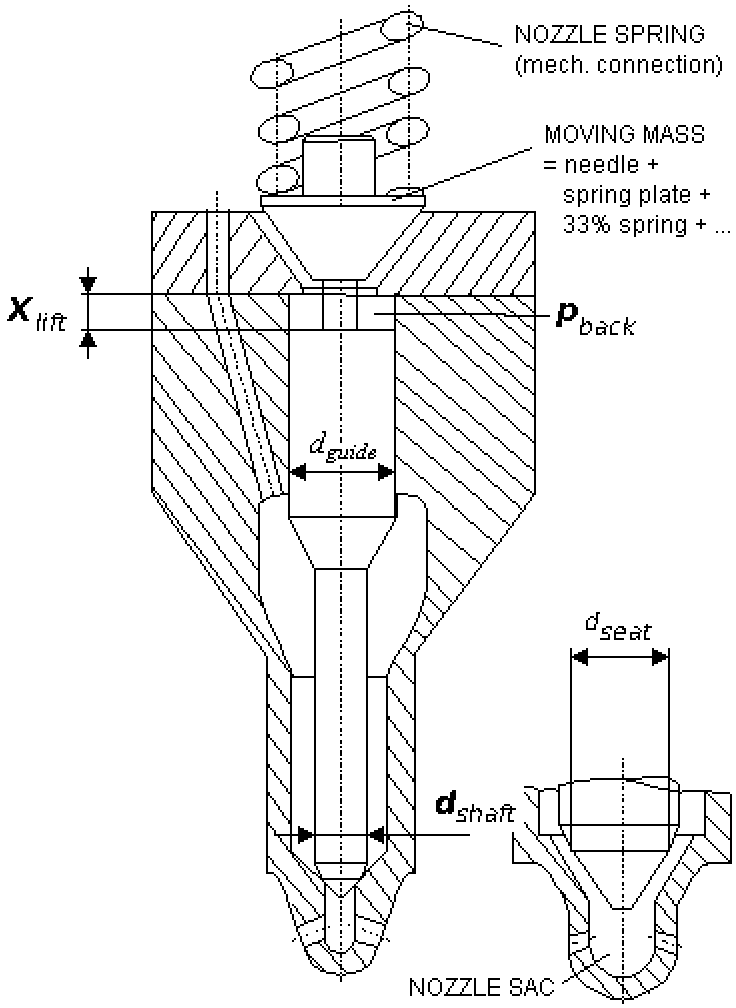


Figure 200: Standard Needle with Nozzle

The cross-section of a **Hollow Needle** valve with large **SAC Nozzle** and **Needle** geometry are shown in Figure 201 and Figure 202.

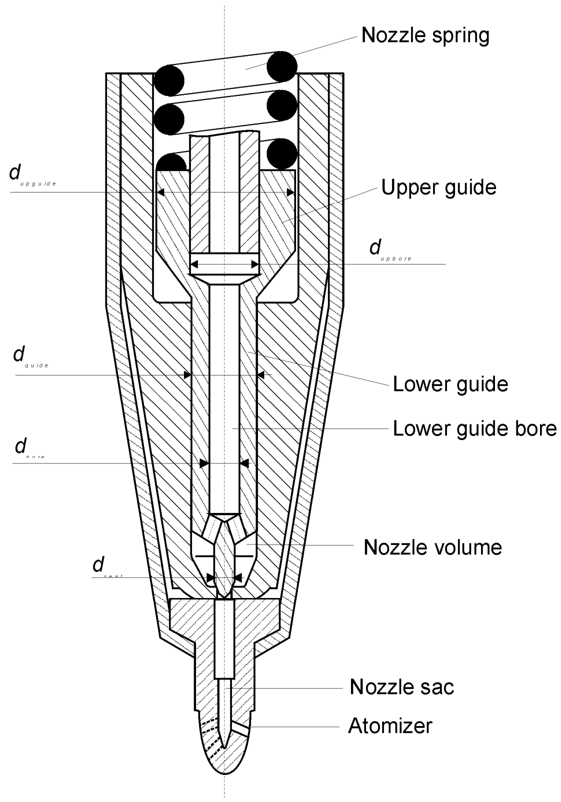


Figure 201: Hollow Needle Valve

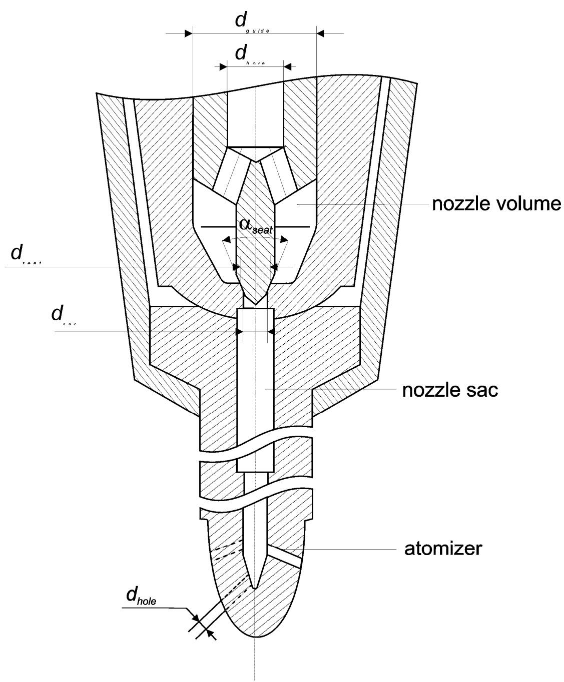
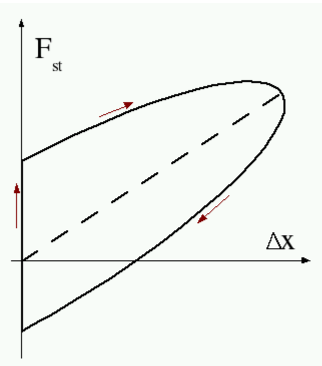


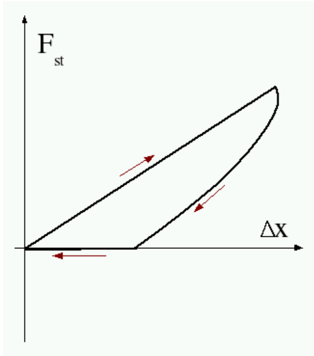
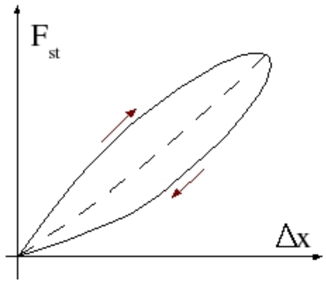
Figure 202: SAC Nozzle with a Hollow Needle

21.1.1. Input Parameters

Description of input data for the **Standard Needle**:

Element name	The name of the element is specified as default.
1 DOF (x)	**Needle** possesses 1 degree-of-freedom (DOF): translation in x-direction.
3 DOF (x,y,z)	**Needle** possesses 3 degrees-of-freedom (DOFs): translations in x, y and z-directions.
Diametral Gap at Guide(s)	Press this bar to open the input dialog for the gap/clearance of needle guide (refer to Section 21.1.2).
Rigid Body	**Rigid Needle** is a rigid body with 1 DOF, i.e. without elasticity between the input and output end. Coordinates and velocities on input and output end of **Needle** element are same.
Moving mass	The moving mass *m* consists of the needle mass, spring plate mass and 33% mass of the connected springs (refer to Figure 200).
Coulomb friction force	Specify constant dry friction force (if any).
Elastic Body	**Elastic Needle** implies an elastic body with the axial elasticity between the input and output end. Stiffness and damping characteristics of **Elastic Needle** are calculated according to the geometric and material data.
Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the elastic needle (refer to **7.1.2**).
Maximum lift of needle	Specify maximum lift of needle X_{lift} (refer to Figure 200). **Note:** If Needle has no mechanical stop, this button has to be switched off. In this case Needle stop data is automatically disabled.
NEEDLE GUIDE/SEAT	
Diameter of needle guide	Specify diameter of the needle guide d_{guide} (refer to Figure 200).
Diameter of needle seat	Specify diameter of the needle seat d_{seat} (refer to Figure 200).
HOLLOW NEEDLE	
Diameter of upper guide bore	Specify diameter of the upper bore of the hollow needle guide d_{upbore} (refer to **Figure 246**). **Note:** If Needle is connected to RSN Throttle element (by special connection), the "Hollow Needle" option is automatically disabled.
SQUEEZING FLUID AT NEEDLE CLOSING	
Damping coefficient	Specify coefficient of velocity-proportional damping for needle closing (viscous damping of squeezing fluid). If it is not specified or set to 0, 'dry' needle motion is considered. If no a-priori estimates are available, this coefficient should be left 0.

NEEDLE SEAT/STOP	
Linear or Pseudo-linear Seat/Stop model	
Needle seat stiffness	Specify the needle seat stiffness on input end (k_{in_st}). Stiffness is active when needle lift $x < 0$.
Needle stop stiffness	Specify the needle stop stiffness on output end (k_{out_st}). Stiffness is active when needle lift $x > X_{lift}$.
Needle seat damping	Specify the needle seat damping on input end (c_{in_st}). Damping is active when needle lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Needle stop damping	Specify the needle stop damping on output end (c_{out_st}). Damping is active when needle lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
Nonlinear Seat/Stop model	
Needle seat linear stiffness	Specify the needle seat linear stiffness on input end (k_{in_st}). Stiffness is active when needle lift $x < 0$.
Needle stop linear stiffness	Specify the needle stop linear stiffness on output end (k_{out_st}). Stiffness is active when needle lift $x > X_{lift}$.
Needle seat Stiffness factor	Specify the needle seat stiffness factor on input end (n_{in_st}).
Needle stop Stiffness factor	Specify the needle stop stiffness factor on output end (n_{out_st}).
Needle seat Elastic impact coef.	Specify the needle seat elastic impact coefficient on input end (α_{in_st}).
Needle stop Elastic impact coef.	Specify the needle stop elastic impact coefficient on output end (α_{out_st}).
Linear Seat/Stop model	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**)  **Figure 203: Linear stop model**

Pseudo-linear Seat/Stop model	This model is derived from the linear model by introducing additional switching function (see equations **65** and **68**). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact.  **Figure 204: Pseudo-linear stop model**
Non-linear Seat/Stop model	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model[38] model is used.  **Figure 205: Non-linear stop model**

[38] K. H. Hunt, F. R. E. Crossley, *Coefficient of Restitution Ineterpreted as Damping in Vibroimpact*, ACME Journal of Applied Mechanics, 1975.

PRESSURE ALONG NEEDLE SEAT	
Constant (basic model)	If this button is active, pressure along needle seat is kept constant (equal to sac pressure acting under the needle seat).
Distributed (advanced model)	If this button is active, a new model for pressure distribution along needle seat is applied (refer to section 21.1.7.1). Dynamic pressure under the needle seat is calculated from the formula: $p_{dyn} = \frac{p_{in} - p_{sac}}{d_{seat} - d_{sac}}(d - d_{sac}) + p_{sac} - \frac{1}{2}\rho\left(\frac{Q}{d\pi X_{lift}\sin(\alpha_{saet})}\right)^2$ where: p_{dyn} - dynamic pressure at diameter d p_{in} - pressure in nozzle chamber p_{sac} - pressure in sac volume d_{seat} - needle seat diameter d_{sac} - diameter of nozzle sac X_{lift} - needle lift α_{seat} - angle at needle seat Q - volumetric flow
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	
NOZZLE HOUSING ELASTICITY	
User-defined	
Geometric and Stiffness Data …	Press this bar to open the input dialog of the geometric and stiffness characteristics of the elastic nozzle housing (refer to section 21.1.3).
Program-calculated	
Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the elastic nozzle housing (refer to section 21.1.3).

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of seat and stop forces (F_{seat}, , F_{stop}).	

21.1.2. Diametral Gap at Guide(s)

For 3DOF needle option, detailed definition of the geometry and clearance of the needle guide(s) is/are required. The subdialog “Diametral Gap of Needle Guide(s)” allows the definition of geometric properties of various needle guides. For the needle model with active 3 DOFs, these properties are required for the calculation of the reaction force(s) at needle contact with seat and guide surfaces as shown in Figure 206.

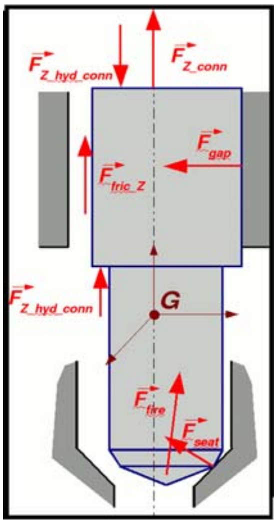
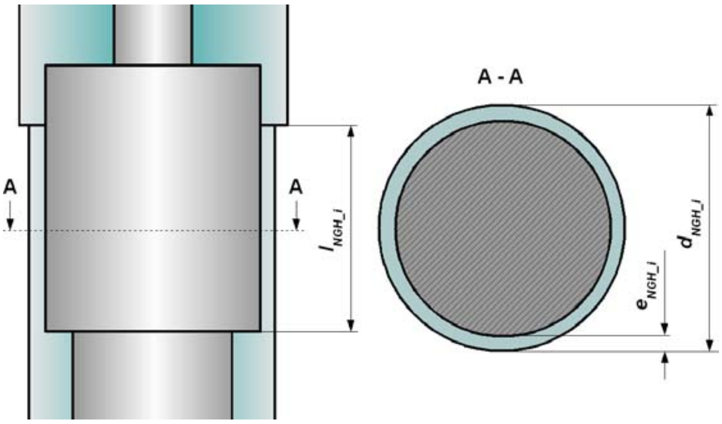
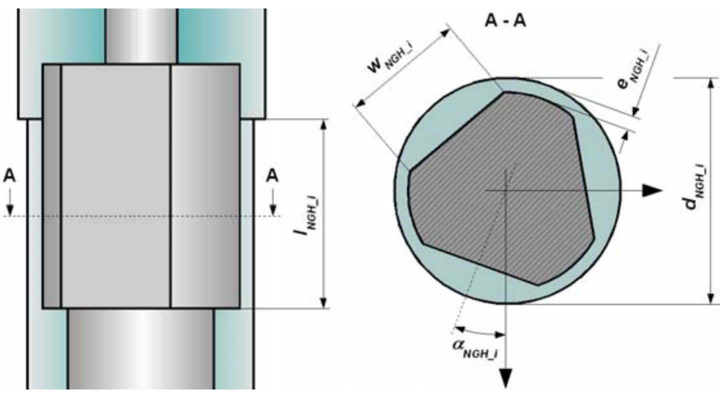
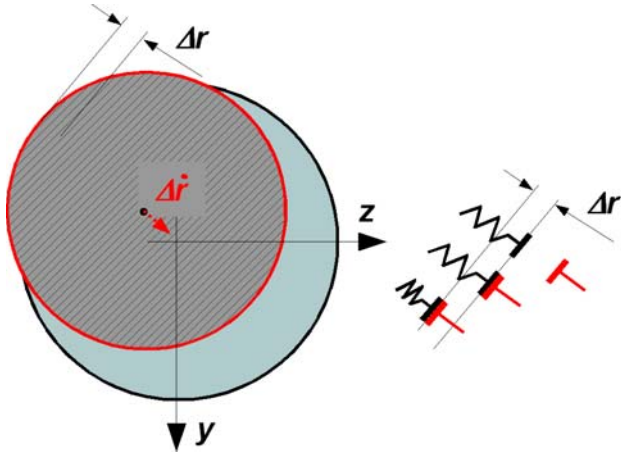


Figure 206: Displaced Needle with Applied Forces

Description of input data:

Gap/Clearance of Needle Guide(s)	
<u>**Regular Cylinder Guide**</u>	
Take from Connected Leakage element(s)	Guide diameter, length and diametral gap (necessary for calculation of needle reaction forces $\boldsymbol{F_{gap}}$) are taken from the connected **Leakage** element.
Specify in Table	Guide diameter, length and diametral gap have to be specified directly in the table. If needle has more than one regular cylinder guide, the data of each guide have to be specified in a separate table row.

	Specify i^{th} guide diameter in first column (d_{NHG_i}). Specify i^{th} guide length in second column (l_{NHG_i}). Specify i^{th} diametral gap in third column (e_{NHG_i}).  **Figure 207: Regular Cylinder Guide**
<u>Cut Cylinder Guide (with symetric segments)</u>	
Specify in Table	Diverse geometric data of Cut Cylinder Guide geometry have to be specified in the table. If needle has more than one cut cylinder guide, the data of each guide have to be defined in a separate table row.
	Specify j^{th} guide diameter in 1st column (d_{NHG_j}). Specify j^{th} guide length in 2nd column (l_{NHG_j}). Specify j^{th} diametral gap in 3rd column (e_{NHG_j}). Specify j^{th} segment width in 4th column (w_{NHG_j}). Specify j^{th} angle to y-axis in 5th column (α_{NHG_j}). Specify j^{th} number of cuts in 6th column (N_{NHG_j}).  **Figure 208: Cut Cylinder Guide with 3 Segments**
<u>Contact Force calculation</u>	
Oil film size (damping)	Specify fuel film size at nozzle wall for the application of viscous damping force.

Nozzle wall stiffness at contact with guide	Contact forces are calculated according to the standard equations as in the needle seat/stop model which may be linear, pseudo-linear or non-linear. Contact stiffness and damping or linear stiffness, stiffness factor and elastic impact coefficient are taken from the needle seat/stop model (see Needle input parameters).  **Figure 209: Overlap between Cylindrical Needle Guide and Nozzle Body at Contact**
Radial wall stiffness	Specfiy the stiffness of the nozzle wall at contact with guide
Radial wall damping	Specfiy the damping of the nozzle wall at contact with guide
Hertz contact model between wall & guide	(refer to Section 21.1.7.2) (Reaction Forces at Needle Cylindrical Guide)
Young's modulus	Specify the Young's modulus of guide material (E_{guide}).
Poisson's ratio	Specify Poisson's ratio of guide material (υ_{guide}).
Contact damping ratio	Specify contact damping ratio (r_C).
☞	Some common rail injector needles possess multiple guides. Among them there are cylindrical guides with cuts or segments (for the free by-pass of the fuel). The reaction force of such a guide is calculated analogously as for the regular cylindrical guide (assuming that the contact with nozzle body occurs always at cylindrical surfaces).

21.1.3. Nozzle Housing Elasticity

Nozzle housing elasticity option allows to calculate axial deformation of nozzle housing at needle seat position due to inner/outer pressure forces acting on the housing. Stiffness and damping constants k and c in Figure 212 represent the axial elasticity and damping of the housing between needle seat and nozzle clamping point (e.g. cylinder head).

Stiffness and damping of the nozzle housing can be specfied by the user (**user-defined** option) or calculated by the program (**program-calculated** option). The latter options requires geometry and material data of the nozzle housing to be specified by the user.

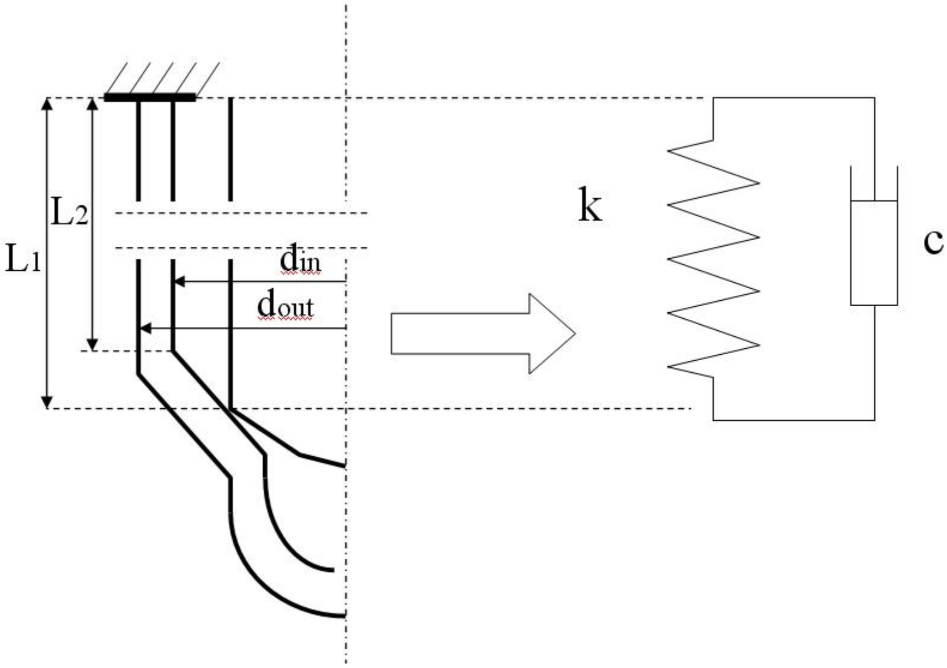


Figure 210: Mechanical model of elastic nozzle housing

Description of input data:

USER-DEFINED	
Stiffness Data	
Stiffness	Specify the stiffness at needle seat point.
Damping	Specify the damping at needle seat point.
Geometric Data	
Inner diameter	Specify the inner nozzle diameter.
Outer diameter	Specify the outer nozzle diameter (optional).
PROGRAM-CALCULATED	
Axial nozzle housing stiffness k is calculated from the stiffness of each cylindrical section (refer to Figure 211): $$\frac{1}{k} = \sum_i \frac{1}{k_i} \quad \text{where} \quad k_i = \frac{EA_i}{L_i}$$ Cross-sectional area of *k-th* nozzle housing section is given by: $$A^k = \pi \frac{\left(d_{out}^k\right)^2 - \left(d_{in}^k\right)^2}{4}$$ Nozzle housing damping c is calculated from the critical damping c_{crit}: $$c_{crit} = 2\sqrt{mk}\ ,\ c = r_c c_{crit}$$ where r_c is equivalent viscous damping ratio and m is housing mass calculated automatically from the total volume of all sections and material density.	

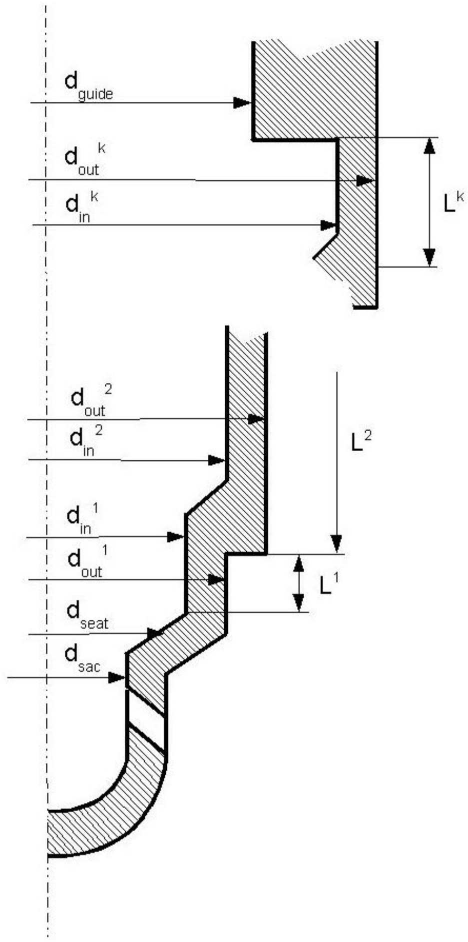


Figure 211: Nozzle housing schematic with multiple sections

Material Data	
Solid Properties …	Press this bar to define local solid properties (refer to **7.1.2**). Global solid properties are active by default.
Equivalent viscous damping ratio	Specify equivalent viscous damping ratio.
Geometric Data	
Cross-section type:	
Circular	Geometry of elastic nozzle housing with circular cross-sections is shown in Figure 211.
Cross-section table	Specify inner/outer diameters D_i and lengths L_i of all nozzle housing sections.
Other	Elastic nozzle housing with other (non-circular) cross-sections requires input of cross-sectional areas instead of diameters.
Cross-section table	Specify inner/outer areas A_i and lengths L_i of all nozzle housing sections.

21.1.4. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial conditions:

X-coordinate	
input x	Specify the initial position of needle input end in local x-direction.
output x	Specify the initial position of needle output end in local x-direction.
	Note: If **Rigid Body** option is active, then initial coordinates at input and output end must be the same.
Velocity in x-direction	
input v	Specify the initial velocity of needle input end in local x-direction.
output v	Specify the initial velocity of needle output end in local x-direction.
	Note: If **Rigid Body** option is active, then initial velocities at input and output end must be the same.
Y and Z-coordinates (for 3DOF needle only)	
Y	Specify the initial position of needle in local y-direction.
Z	Specify the initial position of needle in local z-direction.
Velocity in y and z directions for 3DOF needle only)	
y	Specify the initial velocity of needle in local y-direction.
Z	Specify the initial velocity of needle in local z-direction..

All initial values that are not specified under initials are set to 0.

21.1.5. Modifiable Parameters

Path: **Element│ Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

Coulomb friction force	SI Unit:	N
squeezing fluid damping at needle closing	SI Unit:	Ns/m
maximum lift of needle	SI Unit:	m

21.1.6. Output Parameters

Path: **Element| Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

x-coordinate of needle on input end	Local x-coordinate (lift) of input end of Needle.
velocity in x-direction on input end	Velocity in local x-direction of input end of Needle.
acceleration in x-direction on input end	Acceleration in local x-direction of input end of Needle.
x-coordinate of needle on output end	Local x-coordinate (lift) of output end of Needle.
velocity in x-direction on output end	Velocity in local x-direction of output end of Needle.
acceleration in x-direction on output end	Acceleration in local x-direction of output end of Needle.
hydraulic force acting on needle	Sum of hydraulic forces acting on the input and output end of Needle in x-direction.
resultant force	Resultant force of needle is the sum of all connection forces and external forces acting on it in x-direction $F_R = m\ddot{x}$
mechanical force	Mechanical force is the sum of the preload forces, stiffness and damping forces, Coulomb friction force and forces from the needle stops in x-direction
shear force from leakage	Shear force is calculated in **Annular gap** element, which is connected to needle element via special connection.
hydraulic force on tip in x-direction	Hydraulic force acting on needle tip (under seat) in local x-direction
Y and Z-directions (for 3DOF needle only)	
y-coordinate of needle	Local y-coordinate (lift) of Needle.
velocity in y-direction	Needle velocity in local y-direction.
acceleration in y-direction	Needle acceleration in local y-direction
hydraulic force on tip in y-direction	Hydraulic force acting on needle tip (under seat) in local y-direction
z-coordinate of needle	Local z-coordinate (lift) of Needle.
velocity in z-direction	Needle velocity in local -direction.
acceleration in z-direction	Needle acceleration in local z-direction
hydraulic force on tip in z-direction	Hydraulic force acting on needle tip (under seat) in local z-direction

needle seat deformation in x-direction	Axial deformation of nozzle housing at needle seat position due to pressure forces (calculated only for active “Nozzle housing elasticity” option)

21.1.7. Forces Acting on Needle

21.1.7.1. Forces on 1-DOF Needle

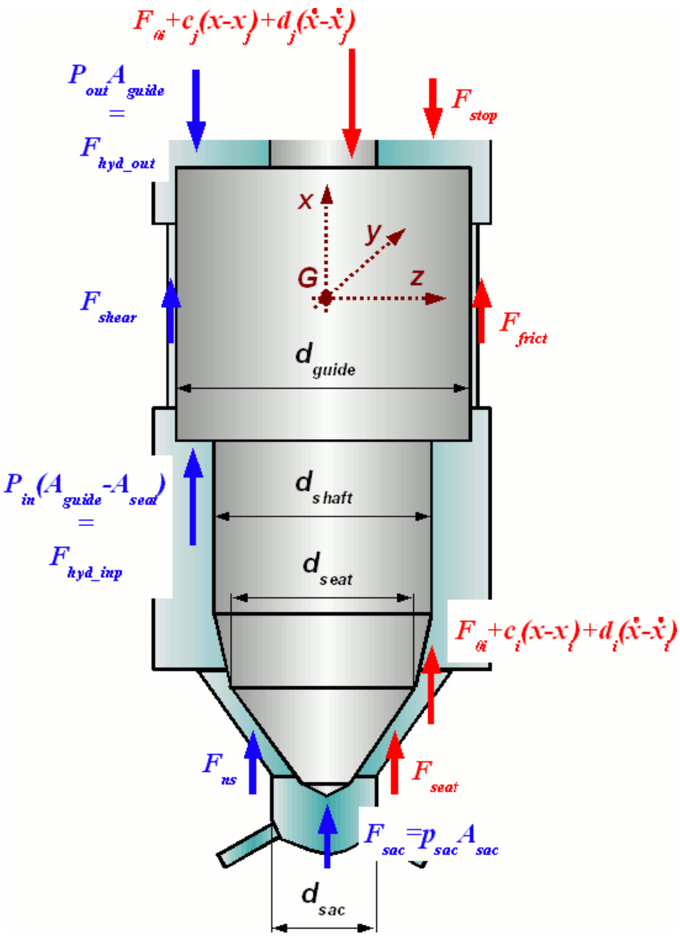


Figure 212: Hydraulic and Mechanical Forces acting on Needle in x-direction

Forces acting on needle body with 1 degree-of-freedom are shown in Figure 212. These forces consist of hydraulic forces from seat and guide(s), mechanical connection (spring) forces, reaction forces from seat and stop, shear forces from leakage(s) and Coulomb friction force (if any). All forces are acting in x-direction. They are listed in the table below.

x	local x-coordinate of needle
Characteristics of connections / Connection Forces /:	
x_i	local x-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
ci ,ki	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end

cj , kj	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
$F0i$,$F0j$	preload forces of the i-th input and j-th output mechanical connections, respectively
Forces on needle are defined in local x-direction:	
F_{hyd_in}	hydraulic force at input end (needle opening force) (pressurized area)
F_{hyd_out}	hydraulic force at output end (pressurized area)
F_{sac}	hydraulic force acting on the needle tip from the sac volume
F_{ns}	hydraulic force acting on the needle between its seat and sac volume area
F_{shear}	viscous friction force due to leakage (if any)(calculated in Leakage element)
F_{frict}	Coulomb friction force
F_{damp}	damping force from squeezing fluid at needle closing
F_{seat}, F_{stop}	additional forces from needle seat (input end) and stop (output end)

Hydraulic Forces

For all **Needle** types excluding the RSN Needle, the opening force F_{hyd_in} consists of two components:

$$F_{hyd\_in} = F_{guide} + F_{tip}, \tag{412}$$

where F_{guide} is the hydraulic force acting upon needle from the nozzle volume (under needle guide) and F_{tip} is the hydraulic force acting on the needle tip.

Hydraulic force F_{guide} on the **Standard Needle** guide is calculated from the equation:

$$F_{guide} = p_{in}\left(A_{guide} - A_{seat}\right) = \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{seat}^2\right), \tag{413}$$

where p_{in} is the input pressure (in nozzle volume), A_{guide} and A_{seat} are the cross-sectional areas of the needle guide and seat and d_{guide} and d_{seat} are the diameters of the needle guide and seat, respectively.

For **Hollow Needle**, the guide hydraulic force F_{guide} is given by:

$$F_{guide} = \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{upbore}^2 - d_{seat}^2\right), \tag{414}$$

where d_{guide}, d_{upbore} and d_{seat} are the diameters of the lower needle guide, upper guide bore and needle seat, respectively.

For **RSN Needle** with a collar (connected by a special connection to **RSN Throttle** element), needle opening force F_{hyd_in} consists of three components:

$$F_{hyd\_in} = F_{guide} + F_{cl} + F_{tip}, \tag{415}$$

where F_{cl} is the hydraulic force acting on the needle collar (refer to Chapter 20.5.6). Guide hydraulic force F_{guide} is calculated from the equation:

$$F_{hyd\_out} = p_{in}\left(A_{guide} - A_{cl}\right) = \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{cl}^2\right), \tag{416}$$

where A_{cl} is the cross-sectional area and d_{cl} is the diameter of the needle collar defined in the input dialog of **RSN Throttle**.

Needle tip force F_{tip} of the **Standard Needle** model is calculated from the formula:

$$F_{tip} = F_{sac} + F_{ns}, \tag{417}$$

where F_{sac} is the hydraulic force acting on the needle tip from the sac volume:

$$F_{sac} = p_{sac} A_{sac} = \frac{\pi}{4} p_{sac} d_{sac}^2, \tag{418}$$

and F_{ns} is the hydraulic force acting on the needle between its seat and sac volume area. For the constant pressure along needle seat it is calculated from the equation:

$$F_{ns} = p_{sac}\left(A_{saet} - A_{sac}\right) = \frac{\pi}{4} p_{sac}\left(d_{seat}^2 - d_{sac}^2\right). \tag{419}$$

For the distributed pressure along needle seat, force F_{ns} is a function of the nozzle geometry, needle lift, pressure in nozzle and sac volumes and flow rate:

$$F_{ns} = f\left(geometry, x_{lift}, p_{in}, p_{sac}, \dot{Q}\right). \tag{420}$$

For the more detailed explanation see Pressure Distribution along Needle Seat below.

Pressure p_{sac} is the pressure in the nozzle sac (for a **SAC Nozzle**) or pressure in the area before the spray holes (for a **VCO Nozzle**).

For all **Needle** types excluding the hollow needle, hydraulic force on the output end of needle F_{hyd_out} is calculated from the formula:

$$F_{hyd\_out} = p_{out} A_{guide} = \frac{\pi}{4} p_{out} d_{guide}^2, \tag{421}$$

where p_{out} is the pressure in the chamber above the needle guide (nozzle outlet pressure).

Hydraulic force on the output end of **Hollow Needle** F_{hyd_out} is calculated from:

$$F_{hyd\_out} = \frac{\pi}{4} p_{out}\left(d_{guide}^2 - d_{upbore}^2\right). \tag{422}$$

Damping force from squeezing fluid at needle closing F_{damp} is defined as follows:

$$\begin{aligned} &\forall\ \dot{x} < 0 : F_{damp} = c_{fluid}\dot{x}, \\ &\forall\ \dot{x} \geq 0 : F_{damp} = 0, \end{aligned} \tag{423}$$

where c_{fluid} is the damping coefficient of squeezing fluid at needle seat. Clearly, F_{damp} is always zero at needle opening (upward motion out of seat) and, if $c_{fluid} \neq 0$, nonzero at needle closing (downward motion to seat).

Seat/Stop Reaction Forces

Reaction Forces with Rigid Nozzle Housing

Rigid nozzle housing case implies active **Seat/Stop contact model** option and inactive **Nozzle housing elasticity** option. In this the reaction force from **Needle** seat F_{seat} is defined as follows:

Linear stop model

$$\begin{aligned} &\forall\, x < 0 : F_{seat} = c_{seat}\dot{x} + k_{seat}x\,, \\ &\forall\, x \geq 0 : F_{seat} = 0, \end{aligned} \tag{424}$$

Pseudo-linear stop model

$$\begin{aligned} &\forall x < 0 \\ &\qquad \forall \dot{x} < 0 \quad : \quad F_{seat} = k_{seat}x\,, \\ &\qquad \forall \dot{x} \geq 0 \quad : \quad F_{seat} = c_{seat}\dot{x} + k_{seat}x\,, \\ &\qquad \forall F_{seat} > 0 \quad : \quad F_{seat} = 0\,, \\ &\forall x \geq 0 \\ &\qquad\qquad : \quad F_{seat} = 0\,, \end{aligned} \tag{425}$$

where c_{seat} and k_{seat} are the damping and stiffness coefficients of the needle seat at its contact with the needle tip (input end).

Similarly, force from needle stop F_{stop} is given by:

Linear stop model

$$\begin{aligned} &\forall\, x \leq X_{lift} : F_{stop} = 0\,, \\ &\forall\, x > X_{lift} : F_{stop} = c_{stop}\dot{x} + k_{stop}\left(x - X_{lift}\right), \end{aligned} \tag{426}$$

Pseudo-linear stop model

$$\begin{aligned} &\forall x \leq X_{lift} \\ &\qquad\qquad : \quad F_{stop} = 0\,, \\ &\forall x > X_{lift} \\ &\qquad \forall \dot{x} > 0 \quad : \quad F_{stop} = k_{stop}\left(x - X_{lift}\right), \\ &\qquad \forall \dot{x} \leq 0 \quad : \quad F_{stop} = c_{stop}\dot{x} + k_{stop}\left(x - X_{lift}\right), \\ &\qquad \forall \mathrm{F}_{stop} < 0 \quad : \quad F_{stop} = 0\,, \end{aligned} \tag{427}$$

where c_{stop} and k_{stop} are the damping and stiffness constants of the needle stop at its contact with the needle guide (output end) and X_{lift} is the maximum needle lift.

Reaction Forces with Elastic Nozzle Housing

Elastic nozzle housing case implies active **Nozzle housing elasticity** option. **Seat/Stop contact model** option can be either active (elastic contact) or inactive (rigid contact). Generally in this case we have to consider both the force at needle contact with its seat and the force resulting from the elastic deformation of the nozzle body. This deformation is caused by the fluid pressure oscillations in the nozzle chamber (refer to section 21.1.9).

Here we have to distinguish between the absolute and relative needle lift:

- x_{abs} - needle tip motion in absolute (non-moving) coordinate system associated with the clamping point of nozzle/injector body (e.g. cylinder head)
- x_{rel} - needle tip motion in relative (moving) coordinate system associated with the needle seat
- δ_{seat} - deflection of nozzle housing at its seat position (i.e. motion of needle seat in absolute coordinate system) in x direction

Relative and absolute needle motion are related by a simple equation:

$$x_{rel} = x_{abs} - \delta_{seat}\cos(\pi) = x_{abs} + \delta_{seat}\,. \tag{428}$$

A necessary and sufficient condition for needle contact with the seat is: $x_{rel} \le 0$.

If **Seat/Stop contact model** option is inactive (“rigid” contact), then reaction force on **Standard Needle** tip from the seat F_{seat} is given by:

$$\begin{aligned} &\forall\, x_{rel} < 0 : F_{seat} = c_{body}\dot{x}_{abs} + k_{body}x_{abs}\,, \\ &\forall\, x_{rel} \ge 0 : F_{seat} = 0, \end{aligned} \tag{429}$$

where c_{body} and k_{body} are the damping and stiffness coefficients of the nozzle body.

If **Seat/Stop contact model** option is active (elastic contact), then reaction force on **Standard Needle** tip F_{seat} is defined as:

Linear stop model

$$\begin{aligned} &\forall\, x_{rel} < 0 : F_{seat} = c_{sum}\dot{x}_{abs} + k_{sum}x_{abs}\,, \\ &\forall\, x_{rel} \ge 0 : F_{seat} = 0, \end{aligned} \tag{430}$$

Pseudo-linear stop model

$$\begin{array}{lll} \forall x_{rel} < 0 & & \\ \quad \forall \dot{x}_{rel} < 0 & : & F_{seat} = k_{sum}x\,, \\ \quad \forall \dot{x}_{rel} \ge 0 & : & F_{seat} = c_{sum}\dot{x} + k_{sum}x\,, \\ \quad \forall F_{seat} > 0 & : & F_{seat} = 0\,, \\ \forall x_{rel} \ge 0 & & \\ & : & F_{seat} = 0\,, \end{array} \tag{431}$$

where c_{sum} and k_{sum} are the resultant damping and stiffness coefficients given by:

$$\frac{1}{c_{sum}} = \frac{1}{c_{body}} + \frac{1}{c_{seat}}, \quad \frac{1}{k_{sum}} = \frac{1}{k_{body}} + \frac{1}{k_{seat}}. \tag{432}$$

Reaction force from needle stop F_{stop} is calculated in the same way as for rigid nozzle housing (irrelevant of the **Nozzle housing elasticity** option).

Needle Spring Force

Needle spring has to be defined as an output mechanical connection from the Needle to a **Mechanical** or **Hydromechanical Boundary** element. The preload force of the nozzle spring has to be calculated from the nozzle cracking (opening) pressure:

$$F_0 = \frac{\pi}{4} p_{crack}\left(d_{guide}^2 - d_{seat}^2\right) + \frac{\pi}{4} p_{cyl} d_{seat}^2, \tag{433}$$

where p_{crack} is the cracking pressure and p_{cyl} is the cylinder pressure at needle opening instant. For high pressure injection systems, the second term can be neglected because $p_{cyl} << p_{crack}$.

The preload force of the **Hollow Needle** spring has to be calculated from the formula:

$$F_0 = \frac{\pi}{4} p_{crack}\left(d_{guide}^2 - d_{upbore}^2 - d_{seat}^2\right). \tag{434}$$

Pressure Distribution along Needle Seat

According to the continuity equation, volumetric flow rate Q through needle seat is constant in space, i.e. along the needle symmetry axis. However, the flow velocity v is variable in time and space. It depends on the cross-sectional flow area under needle seat as shown in Figure 213.

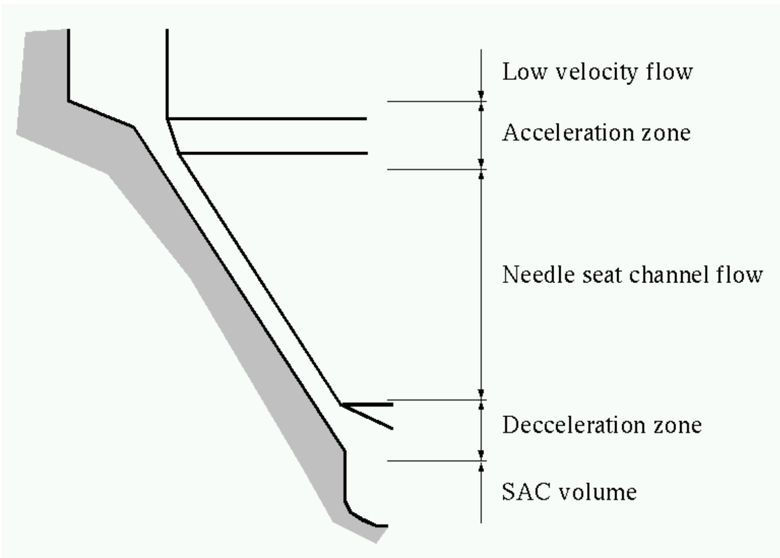


Figure 213: Flow Zones in Sac Nozzle

The pressure distribution model is based on the assumption that nozzle flow passage has a conical shape and that the Bernoulli equation is applicable there:

$$p + \frac{1}{2}\rho v^2 = const. \tag{435}$$

where p is the static pressure and v is the flow velocity ($v = \frac{\dot{Q}}{A}$). Obviously, flow area along needle seat depends on needle lift X_{lift} and needle tip radius r (refer to Figure 214 and Figure 215).

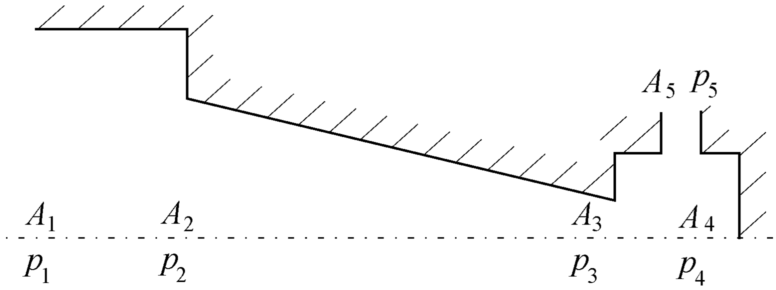


Figure 214: Change of Flow Area inside Nozzle

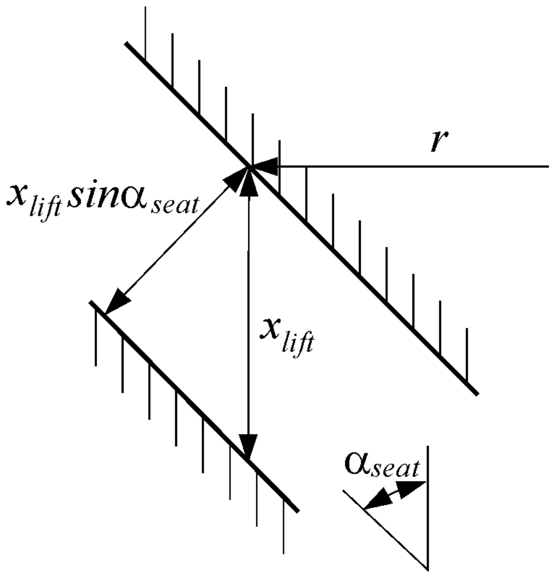


Figure 215: Flow Passage Characteristics at Needle Seat

Static pressure along needle seat is calculated using linear interpolation between nozzle volume pressure and sac pressure:

$$p_{stat} = \frac{p_{in} - p_{sac}}{r_{seat} - r_{sac}}(r - r_{sac}) + p_{sac}. \tag{436}$$

Dynamic pressure along needle seat is determined from the equation:

$$p_{dyn} = p_{stat} - \frac{1}{2}\rho v^2 = p_{stat} - \frac{1}{2}\rho\left(\frac{\dot{Q}}{2r\pi X_{lift}\sin\alpha_{seat}}\right). \tag{437}$$

According to Equation 437, dynamic pressure under needle seat is always smaller than the static pressure as illustrated in Figure 216 and Figure 217.

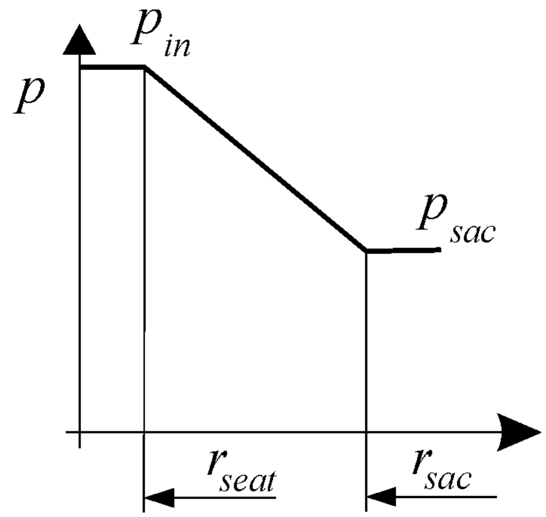


Figure 216: Static Pressure along Needle Seat

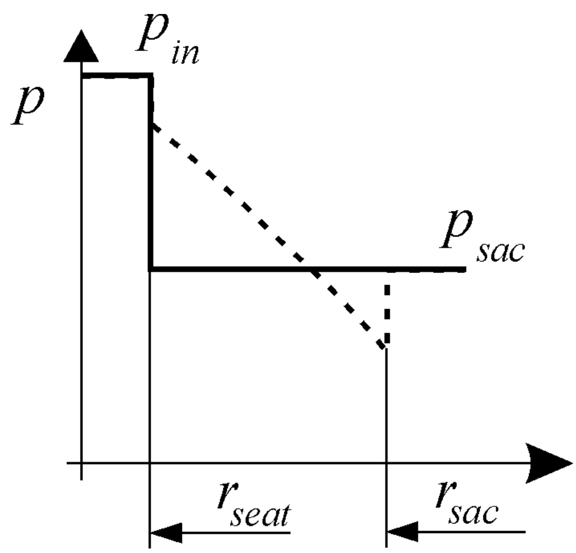


Figure 217: Dynamic Pressure along Needle Seat

To calculate hydraulic force F_{ns} acting on the needle between its seat and sac volume area, the following integral has to be solved:

$$F_{ns} = \int_{r_{sac}}^{r_{seat}} p_{dyn} 2r\pi dr \,. \tag{438}$$

Inserting Equation 437 into Equation 438 and performing integration, we obtain the explicit expression for the hydraulic force F_{ns}:

$$F_{ns} = 2\pi \left[\frac{1}{3} K \left(r_{seat}^3 - r_{tip}^3 \right) + \frac{1}{2} \left(p_{sac} - K r_{tip} \right) \left(r_{seat}^2 - r_{tip}^2 \right) - A \ln \frac{r_{seat}}{r_{tip}} \right], \tag{439}$$

where

$$K = \frac{p_{in} - p_{sac}}{r_{seat} - r_{tip}} \text{ and } A = \frac{1}{2}\rho\left(\frac{\dot{Q}}{2\pi X_{lift}\sin\alpha_{seat}}\right)^2. \text{ If } F_{ns} < 0 \text{ set } F_{ns} = 0.$$

The influence of laminar boundary layer is assumed to be negligible except when needle lift is very small. For this case (laminar flow regime), Equation 437 for p_{dyn} calculation is used in a modified form. During the laminar regime Reynolds number is controlled. According to its ratio to the Reynolds number of turbulent flow start the damping coefficient is calculated. It is used for the correction of the static pressure p_{stat} and flow rate Q in Equation 437.

Figure 218 shows the measured pressure distribution in the flow passage of a typical sac nozzle. Good correlation with the calculation results using Equation 437 has been achieved.

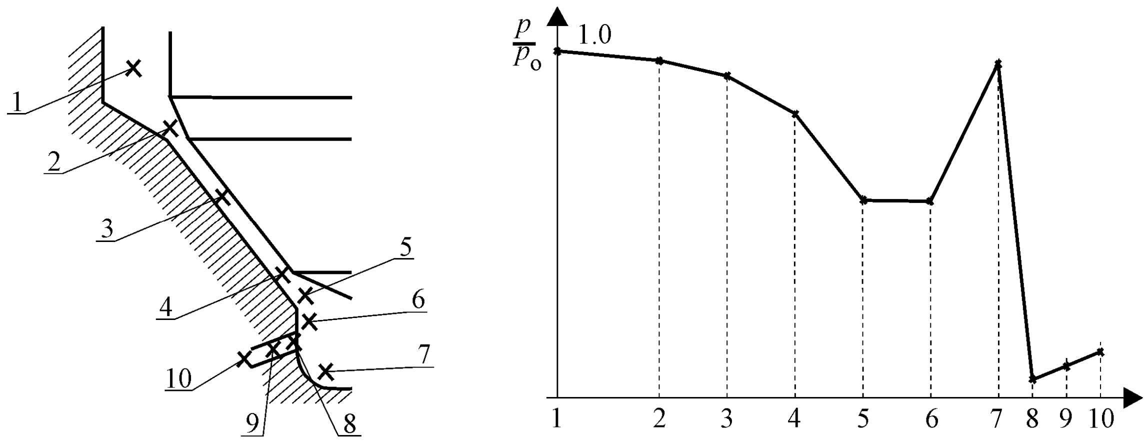


Figure 218: Measured Pressure Distribution in Sac Nozzle

21.1.7.2. Forces on 3-DOF Needle

Forces acting on the needle body with 3 degrees-of-freedom are shown in Figure 219. These forces consist of hydraulic forces from seat and guide(s), mechanical connection (spring) forces, reaction forces from seat, stop and guide(s), shear forces from leakage(s) and Coulomb friction force (if any). Reaction forces at needle seat and guide are now three-dimensional vectors listed below.

Reaction forces on needle in local x, y and z-directions:	
F_{seat} *(x,y,z)*	Reaction force at needle seat (input end)
F_{react_guide} *(x,y,z)*	Reaction forces at needle guide

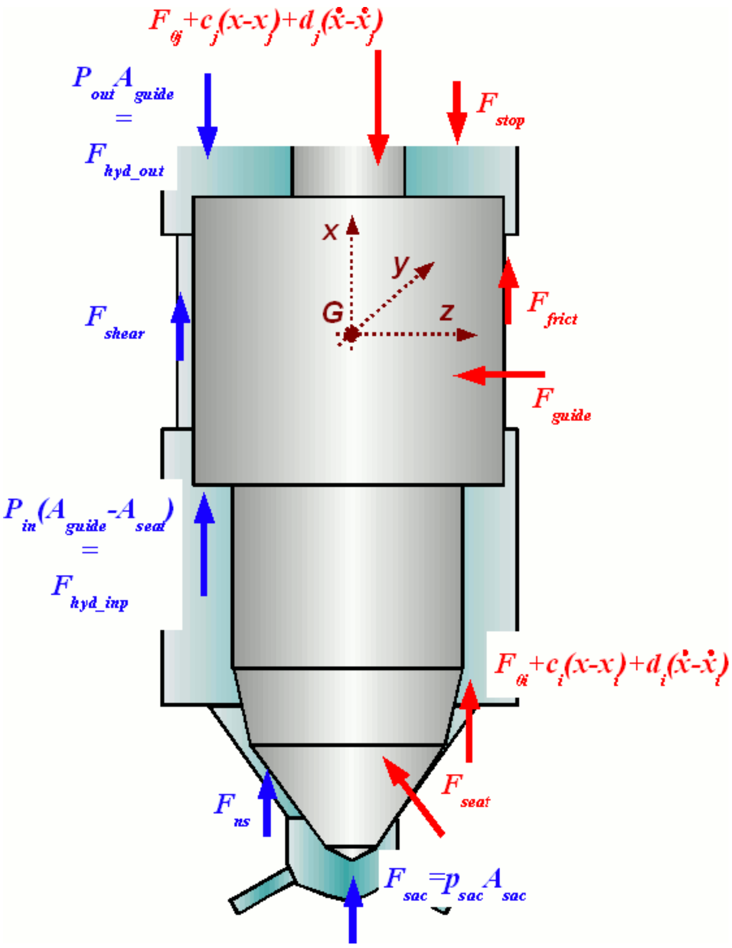


Figure 219: Forces acting on Needle (blue – hydraulic, red – mechanical)

<u>**Reaction Forces at Needle Seat**</u>

The mechanical contact between the needle tip (red contour) and nozzle body (blue contour) is schematically illustrated in Figure 220. The seat reaction force cannot be calculated using Hertz theory because the needle contacts the nozzle body by a sharp circle (there is no finite radius of curvature).

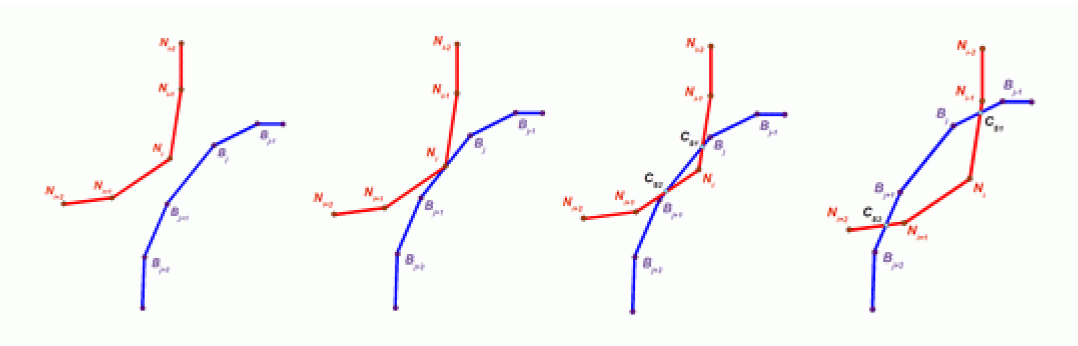


Figure 220: Needle Tip Contour (red) Interfering with Nozzle Body Contour (blue)

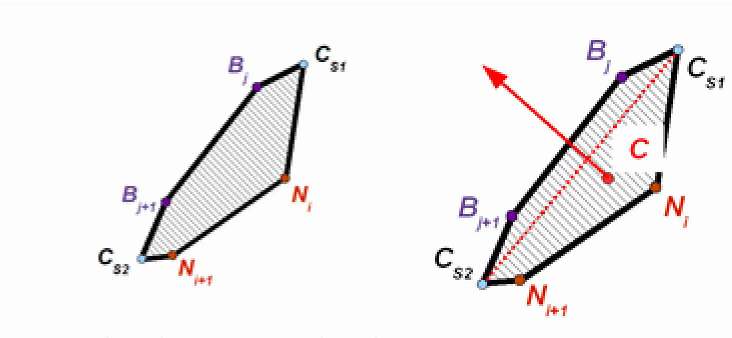


Figure 221: . Overlap Area and Reaction Force at Needle-Nozzle Body Contact

The seat force calculation algorithm consists of the following steps (refer to Figure 220 and Figure 221):

- Check if there is an overlap between two contours: if yes, find the coordinates of the two points at which needle tip intersects the nozzle body.
- Check if there are any edge points on needle tip contour between the intersection points: if so, store their coordinates.
- Check if there are any edge points on nozzle body contour between the intersection points : if so, store their coordinates.
- All above edge points define a polygon. Its area is calculated by the formula

$$A_{pol} = \frac{1}{2}\sum_{i=1}^{N-1}\left(x_i y_{i+1} - x_{i+1} y_i\right), \tag{440}$$

where x_i and y_i are the edge point coordinates in the cross-section plane and N is the number of points.

- Coordinates of the polygon area center can be calculated from:

$$c_x = \frac{1}{6A_{pol}}\sum_{i=1}^{N-1}\left(x_i + x_{i+1}\right)\left(x_i y_{i+1} - x_{i+1} y_i\right),$$
$$c_y = \frac{1}{6A_{pol}}\sum_{i=1}^{N-1}\left(y_i + y_{i+1}\right)\left(x_i y_{i+1} - x_{i+1} y_i\right). \tag{441}$$

- Perpendicular projection from the area center to the line connecting the intersection points can be defined by

$$\lambda = \frac{\left(c_x - s_{1x}\right)\left(s_{2x} - s_{1x}\right) + \left(c_y - s_{1y}\right)\left(s_{2y} - s_{1y}\right)}{\left(s_{2x} - s_{1x}\right)^2 + \left(s_{2y} - s_{1y}\right)^2}, \tag{442}$$

$$p_x = \left(1 - \lambda\right)s_{1x} + \lambda s_{2x},$$
$$p_y = \left(1 - \lambda\right)s_{1y} + \lambda s_{2y}.$$

- Direction of reaction force is calculated from

$$\text{if } p_x < c_x \text{, then } \vec{\mathrm{F}}_\mathrm{c} = \left|\vec{\mathrm{F}}_\mathrm{c}\right| \frac{(p_x - c_x)\vec{i}_x + (p_y - c_y)\vec{i}_y}{\sqrt{(p_x - c_x)^2 + (p_y - c_y)^2}}, \tag{443}$$

$$\text{if } p_x > c_x \text{, then } \vec{\mathrm{F}}_\mathrm{c} = \left|\vec{\mathrm{F}}_\mathrm{c}\right| \frac{(c_x - p_x)\vec{i}_x + (c_y - p_y)\vec{i}_y}{\sqrt{(p_x - c_x)^2 + (p_y - c_y)^2}}$$

- Distance between the intersection points

$$dist_{cs} = \sqrt{(s_{2x} - s_{1x})^2 + (s_{2y} - s_{1y})^2} \tag{444}$$

- Equivalent overlap depth

$$d_{eq} = \frac{A_{pol}}{dist_{cs}} \tag{445}$$

- Contact force magnitude

$$\left|\vec{\mathrm{F}}_\mathrm{seat}\right| = k_c d_{eq} + c_c \dot{d}_{eq} \,. \tag{446}$$

Reaction Forces on Cylindrical Needle Guide (according to Hertz stress theory)

Usually the needle guide has the shape of a regular cylinder as shown in Figure 209. Reaction force of the nozzle body on the needle can be calculated from Hertz stress theory. Assuming a pure line contact between needle guide (regular cylinder) and nozzle body counterpart (cylindrical socket), we can write an equation for the radial surface deformation or interference distance between them.

$$d = 1.82 \frac{q}{E}\left[1 - \ln\left(0.798(qK\gamma)^{\frac{1}{2}}\right)\right], \tag{447}$$

with $K = \frac{D_1 D_2}{D_2 - D_1}$, $\gamma = \frac{2(1-\upsilon^2)}{E}$, $q = \frac{F_{guide}}{l}$.

Here $\boldsymbol{d}$ is the surface deformation (body overlap), $\boldsymbol{F_{guide}}$ – radial force equally distributed along guide length $\boldsymbol{l}$, $\boldsymbol{D_1}$ and $\boldsymbol{D_2}$ are the diameters of inner (guide) and outer (body) cylinders, respectively, $\boldsymbol{E}$ is the Young's modulus and $\boldsymbol{\upsilon}$ Poisson's ratio at contact.

Introducing abbreviations $A_1 = \frac{1.82}{E}$ and $A_2 = 0.798(K\gamma)^{\frac{1}{2}}$, we arrive at the following equation

$$d = A_1 q\left[1 - \ln\left(A_2 q^{\frac{1}{2}}\right)\right]. \tag{448}$$

Solution of this equation for $\boldsymbol{q}$ can be written in the form:

$$q = \frac{-2}{A_1} \frac{d}{W\left(\frac{-2}{A_1} A_2^2 d \exp(-2)\right)}, \tag{449}$$

where $\boldsymbol{W}$ is the so-called Lambert function of any complex number $z = W(z)e^{W(z)}$. Solution of Lambert function with negative argument requires a series expansion with stability analysis.

21.1.7.3. Forces on Needle with Fire Link connection

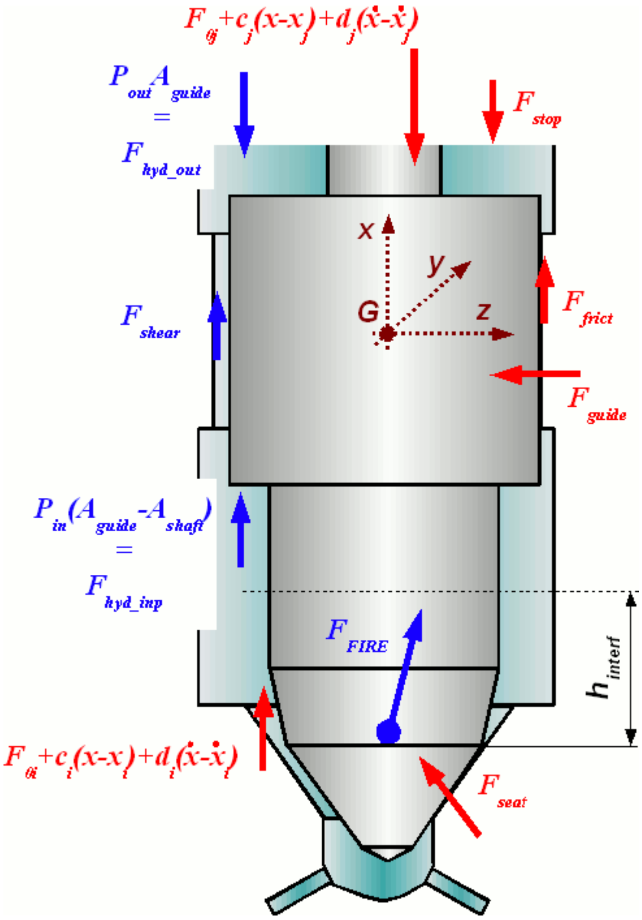


Figure 222: Forces acting on Needle (blue – hydraulic, red – mechanical)

Forces acting on 3-DOF needle connected to the **Fire Link** element are shown in Figure 222. These forces consist of hydraulic forces on guide(s), mechanical connection (spring) forces, reaction forces from seat, stop and guide(s), shear forces from leakage(s), Coulomb friction force (if any) and hydraulic force on seat (external) calculated by Fire multiphase solver.

External forces on needle in local x, y and z-direction:	
F_{FIRE}	Hydraulic force vector calculated by FIRE. It is transferred to Needle model through **FIRE Link (Nozzle)** element during Hydsim-FIRE co-simulation.

Hydraulic Force (calculated in FIRE)

FIRE transfers to BOOST Hydsim the resultant force $\boldsymbol{F} = (\boldsymbol{F}_{Nx}, \boldsymbol{F}_{Ny}, \boldsymbol{F}_{Nz})$ acting on the needle tip up to a user-defined height $\boldsymbol{h}_{interf}$ ($\geq$ **0**). This force results from the three-dimensional pressure distribution in the flow flield around the needle tip contour calculated by FIRE solver as shown in Figure 223.

Note that FIRE force is applied to needle not permanently, but only at active BOOST Hydsim-FIRE data exchange process. Outside of the data exchange range internal BOOST Hydsim seat forces are onset on the needle tip (as if Fire Link does not exist).

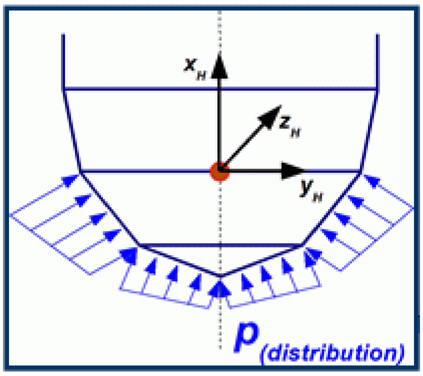


Figure 223: Pressure distribution around needle tip

> ☞ **Note:** **HYDSIM-FIRE Link** is implemented for 1-DOF needle case only. Hence only the x-component of the FIRE force is used in BOOST Hydsim calculation at this stage.

21.1.8. Equations of Motion

21.1.8.1. Standard Needle with Rigid Body

Motion of rigid **Standard Needle** with 1 degree of freedom is governed by the following equations:

$$
\begin{aligned}
& m\ddot{x} \\
& + \sum_{i=1}^{n} c_i \left[\dot{x}\cos\left(\alpha - \alpha_{mc\_i}\right) - \dot{x}_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - \dot{y}_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
& + \sum_{i=1}^{n} k_i \left[x\cos\left(\alpha - \alpha_{mc\_i}\right) - x_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - y_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
& - \sum_{j=1}^{m} c_j \left[\dot{x}\cos\left(\alpha_{mcj} - \alpha\right) - \dot{x}_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + \dot{y}_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right), \\
& - \sum_{j=1}^{m} k_j \left[x\cos\left(\alpha_{mcj} - \alpha\right) - x_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + y_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
& - \sum_{i=1}^{n} F_{0i} \cos\left(\alpha - \alpha_{mc\_i}\right) + \sum_{j=1}^{m} F_{0j} \cos\left(\alpha_{mc\_j} - \alpha\right) \\
& = F_{hyd\_in} - F_{hyd\_out} - F_{frict} - F_{shear} - F_{damp} - F_{seat} - F_{stop}
\end{aligned}
\tag{450}
$$

where:

m	Needle mass
x	local x-coordinate of needle
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends

For **Rigid Needle** with 3 degrees of freedom two additional equations in y and z directions have to be added as shown below:

$$
m\ddot{y} = F_{react\_guide\_Ydir} + F_{seat\_Ydir} + F_{FIRE\_Ydir}\ , \tag{451}
$$

$$m\ddot{z} = F_{react\_guide\_Zdir} + F_{seat\_Zdir} + F_{FIRE\_Zdir}, \tag{452}$$

21.1.8.2. Standard Needle with Elastic Body

Motion of **Standard Needle with Elastic Body** in local coordinate system is governed by the following equations:

$$
\begin{aligned}
&\left(m_{inp} + m_{ad\_inp}\right)\ddot{x}_{inp} \\
&+ \sum_{i=1}^{n} c_i \left[\dot{x}_{inp} \cos\left(\alpha - \alpha_{mc\_i}\right) - \dot{x}_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - \dot{y}_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
&+ \sum_{i=1}^{n} k_i \left[x_{inp} \cos\left(\alpha - \alpha_{mc\_i}\right) - x_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - y_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
&- \sum_{i=1}^{n} F_{0i} \cos\left(\alpha - \alpha_{mc\_i}\right) \\
&= +F_{hyd\_inp} - F_{C\_inp} - F_{damp} - F_{seat} + F_{stiff} + F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
&\left(m_{out} + m_{ad\_out}\right)\ddot{x}_{out} \\
&- \sum_{j=1}^{m} c_j \left[\dot{x}_{out} \cos\left(\alpha_{mcj} - \alpha\right) - \dot{x}_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + \dot{y}_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
&- \sum_{j=1}^{m} k_j \left[x_{out} \cos\left(\alpha_{mcj} - \alpha\right) - x_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + y_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
&+ \sum_{j=1}^{m} F_{0j} \cos\left(\alpha_{mc\_j} - \alpha\right) \\
&= F_{hyd\_out} - F_{C\_out} - F_{stop} - F_{stiff} - F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
F_{stiff} &= k_{ps}\left(x_{inp} - x_{out}\right), \\
F_{damp} &= c_{ps}\left(\dot{x}_{inp} - \dot{x}_{out}\right)
\end{aligned} \tag{453}
$$

21.1.9. Nozzle Body Deformation at Needle Seat

For simple nozzle geometry (refer to figure 204), considering only the cylindrical hollow part, the elastic deformation of nozzle body (housing) under pressure is given by:

$$\delta = \frac{F}{k} = \frac{Fl}{AE},\ l = L_2$$

$$F = F_{nozV} + F_{tip} - F_{out} = \left(A_{body\_in} - A_{seat}\right)p_{nozV} + F_{tip} - A_{body\_out}\,p_{out}$$

$$A = A_{body\_out} - A_{body\_in} = \frac{\pi}{4}\left(d_{out}^2 - d_{in}^2\right)$$

where:

Forces on needle are defined in local x-direction:	
δ_{seat}	nozzle housing deformation at needle seat
δ_i	i-th nozzle housing section deformation
k	nozzle body stiffness
l	characteristic length of nozzle housing (cylinder segment L_2)
l_i	i-th nozzle housing section length
A	nozzle housing cross-section area
A_i	i-th nozzle housing section cross-section area
A_{body_in}	inner nozzle body cross-section area
$A_{body\_in,i}$	i-th inner nozzle body section cross-section area
A_{body_out}	outer nozzle body cross-section area
$A_{body\_out,i}$	i-th outer nozzle body section cross-section area
A_{seat}	needle seat area
A_{sac}	sac area
E	nozzle housing material bulk modulus
F	total force acting on nozzle housing (pressurized area)
F_i	i-th nozzle housing section total force
F_{nozV}	force from nozzle volume pressure
F_{tip}	force from pressure under tip
F_{out}	force from outside pressure (e.g. spray chamber)
F_{ns}	hydraulic force acting on the needle between its seat and sac volume area
p_{nozV}	nozzle volume pressure
p_{out}	outside pressure
p_{sac}	sac volume pressure
d_{in}	inner nozzle housing diameter
d_{in}	outer nozzle housing diameter

For general nozzle geometry with multiple cylindrical sections within nozzle volume (refer to 205), the nozzle body deformation (at seat position) is given by:

$$\delta_{seat} = \sum_{i=1}^{n} \delta_i = \frac{1}{E} \sum_{i=1}^{n} \frac{F_i l_i}{A_i} = \frac{1}{E} \sum_{i=1}^{n} \frac{l_i}{A_i} \sum_{j=2}^{i} F_j ,$$

$$\delta_{seat} = \frac{1}{E} \sum_{i=1}^{n} \frac{l_i}{A_{body\_out,i} - A_{body\_in,i}} \sum_{j=2}^{i} F_j = \sum_{i=1}^{n} \frac{1}{k_i} \sum_{j=2}^{i} F_j$$

For constant (basic) model of pressure distribution along needle seat, section force is given by:

$$F_j = \left(A_{body\_in,j} - A_{body\_in,j-1}\right) p_{nozV} + A_{seat} p_{sac} - \left(A_{body\_out,j} - A_{body\_out,j-1}\right) p_{out}$$
$$j = 2 \Rightarrow A_{body\_in,j-1} = A_{body\_in,1} = A_{seat} ; A_{body\_out,j-1} = A_{body\_out,1} = 0$$

For advanced model of pressure distribution along needle seat, section force is given by:

$$F_j = \left(A_{body\_in,j} - A_{body\_in,j-1}\right) p_{nozV} + A_{sac} p_{sac} + F_{ns} - \left(A_{body\_out,j} - A_{body\_out,j-1}\right) p_{out}$$
$$j = 2 \Rightarrow A_{body\_in,j-1} = A_{body\_in,1} = A_{seat} ; A_{body\_out,j-1} = A_{body\_out,1} = 0$$

21.1.10. Excitation Force on Injector Body

Needle tip impact on nozzle seat (at nozzle closing) generates a reaction force on the injector body (housing). If needle has a mechanical stop, then its impact between with the stop (at nozzle opening) also induces a reaction force that will be transferred via the stop to the surrounding structure. Furthermore, during injection needle continuously presses against the stop and between injections - against seat. Additionally, needle spring also imposes a mechanical force on its support (injector body). Sample time series of these excitation forces are shown in Figure 224.

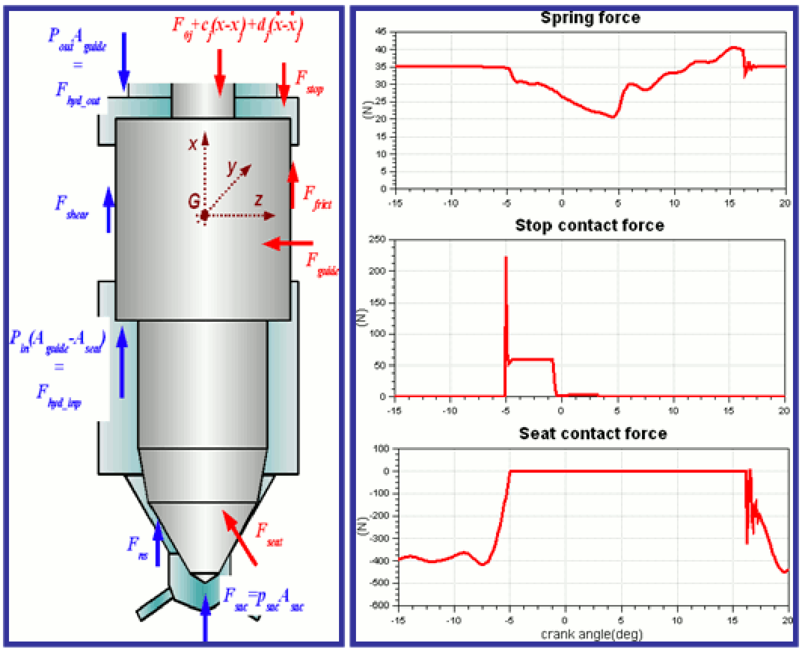
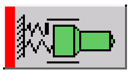
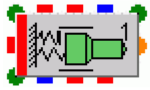


Figure 224: Needle forces acting on the injector housing

.

21.2. Two-spring Needle

Element Name:	**Two-Spring Needle**	
Element Icon:		
Definition:	This dialog defines the properties of a Needle model of a Two-spring injector (TSI) Nozzle.	
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 3 wire pins: 1	

Needle of a **Two-Spring Injector Nozzle** with primary and secondary springs is shown in Figure 225.

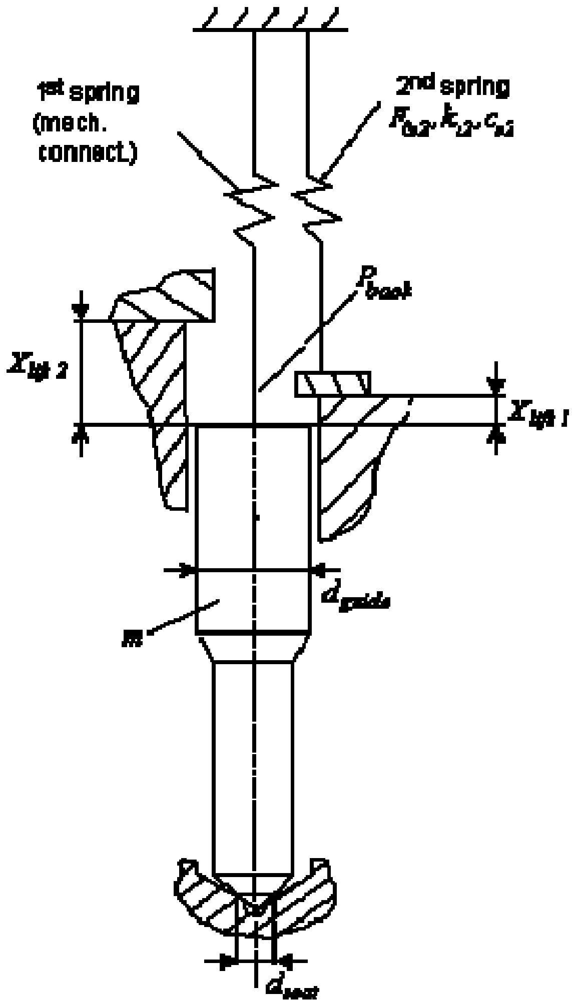


Figure 225: Needle of Two-Spring Injector Nozzle

☞

Note: **Needle** element may have 2 hydraulic and up to 5 mechanical connections (in x direction). These connections are irreversible and have to be specified according to the following rules:
• **Needle** must have an **input** hydraulic connection from the **Nozzle Volume** (high pressure volume under needle guide). **Output** hydraulic connection to the outlet (spring) **Volume** or **Pressure** boundary is optional.
• **Needle primary spring** must be defined as an output mechanical connection to a **Mechanical** (nozzle holder) or **Hydromechanical** boundary (nozzle holder plus outlet pressure). **Secondary spring** data have to be defined directly in the input dialog.
• In case of **Hydromechnical** boundary, mechanical connection is also a hydraulic connection, i.e. it transmits boundary pressure (if any) to **Needle**.

Note: **Up-to-date Needle** model is based on the commonly accepted assumption that at needle opening the nozzle volume pressure is acting until the needle seat and sac pressure is acting under the needle seat. Flow throttling is always effective at needle seat.

Note: For creating a complete nozzle model, **Needle** must be connected to one of the **Nozzle Orifice** elements by a special connection.

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). Needle may have only one hydraulic connection on each end.

Note: Mechanical connections can be defined only in local x-direction.

Note: For Needle element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system, too.

Note: Local x-axis of Needle may be translated and rotated in global coordinate system. Rotation is defined through the parameter: **Direction of Element motion**.

Needle of a **Two-spring Injector Nozzle** with primary and secondary springs is shown in Figure 225.

21.2.1. Input Parameters

The input data of the **Two-spring Needle** is described as follows**:**

Element name	The name of the element is specified as default.
Rigid Body	**Rigid Needle** is a rigid body with 1 DOF, i.e. without elasticity between the input and output end. Coordinates and velocities on input and output end of **Needle** element are same.
Moving mass	The moving mass *m* consists of the needle mass, spring plate mass and 33% mass of the connected springs (refer to Figure 225).

Coulomb friction force	Specify constant dry friction force (if any).
Elastic Body	**Elastic Needle** implies an elastic body with the axial elasticity between the input and output end. Stiffness and damping characteristics of **Elastic Needle** are calculated according to the geometric and material data.
Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the elastic needle (refer to section **7.1.2**).
NEEDLE GUIDE/SEAT	
Diameter of needle guide	Specify the diameter of needle guide d_{guide} (refer to Figure 225).
Diameter of needle seat	Specify the diameter of needle seat d_{seat} (refer to Figure 225).
NEEDLE LIFT	
Maximum primary lift	Primary lift is the distance X_{lift1} which the Needle has to move before the second spring becomes active (refer to Figure 225).
Maximum total lift	Specify maximum total lift of Needle X_{lift2} (refer to Figure 225).
SQUEEZING FLUID AT NEEDLE CLOSING	
Damping coefficient	Specify coefficient of velocity-proportional damping for needle closing (viscous damping of squeezing fluid). If it is not specified or set to 0, 'dry' needle motion is considered. If no a-priori estimates are available, this coefficient should be left 0.
SECONDARY SPRING	
Preload	Specify preload of the Secondary Spring.
Stiffness	Specify stiffness constant of the Secondary Spring.
Damping	Specify viscous damping constant of the Secondary Spring.
NEEDLE SEAT/STOP	
(Linear or Pseudo-linear Stop model)	
Needle seat stiffness	Specify the needle seat stiffness on input end (k_{in_st}). Stiffness is active when needle lift $x < 0$.
Needle stop stiffness	Specify the needle stop stiffness on output end (k_{out_st}). Stiffness is active when needle lift $x > X_{lift}$.
Needle seat damping	Specify the needle seat damping on input end (c_{in_st}). Damping is active when needle lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Needle stop damping	Specify the needle stop damping on output end (c_{out_st}). Damping is active when needle lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
(Nonlinear Stop model)	
Needle seat linear stiffness	Specify the needle seat linear stiffness on input end (k_{in_st}). Stiffness is active when needle lift $x < 0$.

Needle stop linear stiffness	Specify the needle stop linear stiffness on output end (k_{out_st}). Stiffness is active when needle lift $x > X_{lift}$.
Needle seat Stiffness factor	Specify the needle seat stiffness factor on input end (n_{in_st}).
Needle stop Stiffness factor	Specify the needle stop stiffness factor on output end (n_{out_st}).
Needle seat Elastic impact coef.	Specify the needle seat elastic impact coefficient on input end (α_{in_st}).
Needle stop Elastic impact coef.	Specify the needle stop elastic impact coefficient on output end (α_{out_st}).
Seat/Stop model is linear	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**) (refer to **Figure 54**).
Seat/Stop model is pseudo-linear	This model is derived from the linear model by introducing additional switching function (see equations **65** and **68**). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact (refer to **Figure 55**).
Stop model is non-linear	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model[39] model is used.
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$, where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction. Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).
y(e2)	
z(e3)	

[39] K. H. Hunt, F. R. E. Crossley, *Coefficient of Restitution Ineterpreted as Damping in Vibroimpact*, ACME Journal of Applied Mechanics, 1975.

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section 0).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)
Element reaction force is the sum of seat and stop forces (F_{seat}, , F_{stop}) and second spring force ($c_{s2}\dot{x} + k_{s2}x - F_{0s2}$) (if second spring is active).	

21.2.2. Initial Conditions

Path: **Element | Initial Conditions**

Description of initial conditions:

X-coordinate	
input x	Specify the initial position of needle input end in local x-direction.
output x	Specify the initial position of needle output end in local x-direction.
	Note: If **Rigid Body** option is active, then initial coordinates at input and output end must be the same.
Velocity in x-direction	
input v	Specify the initial velocity of needle input end in local x-direction.
output v	Specify the initial velocity of needle output end in local x-direction.
	Note: If **Rigid Body** option is active, then initial velocities at input and output end must be the same.

All initial values that are not specified under initials are set to 0.

21.2.3. Modify Parameter

Path: **Element | Modify**

To activate modification, the option box to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

Coulomb friction force	SI Unit:	N
squeezing fluid damping at needle closing	SI Unit:	---
primary maximum lift of needle	SI Unit:	m
total maximum lift of needle	SI Unit:	m
preload of secondary spring	SI Unit:	N

stiffness of secondary spring	SI Unit:	N/m
damping of secondary spring	SI Unit:	Ns/m

21.2.4. Output Parameters

Output Parameters of **Two-spring Needle** are identical to the **Output Parameters Dialog** of the **Standard Needle** except that there are no outputs in y and z directions (because **Two-spring Needle** is a 1DOF body).

21.2.5. Equation of Motion for Rigid Needle

Motion of **Two-spring Needle** with Rigid Body can be divided into two phases corresponding to primary and secondary needle lift. For the Needle lift less or equal than primary lift, the motion of Needle in local coordinate system is governed by the same equation as for the **Standard Needle** except that there is no force from the output stop:

$$
\begin{aligned}
&\forall x \le X_{lift1}: \\
&m\ddot{x} \\
&+\sum_{i=1}^{n} c_i\left[\dot{x}\cos\left(\alpha-\alpha_{mc\_i}\right)-\dot{x}_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-\dot{y}_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&+\sum_{i=1}^{n} k_i\left[x\cos\left(\alpha-\alpha_{mc\_i}\right)-x_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-y_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&-\sum_{j=1}^{m} c_j\left[\dot{x}\cos\left(\alpha_{mcj}-\alpha\right)-\dot{x}_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+\dot{y}_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right), \qquad (454) \\
&-\sum_{j=1}^{m} k_j\left[x\cos\left(\alpha_{mcj}-\alpha\right)-x_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+y_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&-\sum_{i=1}^{n} F_{0i}\cos\left(\alpha-\alpha_{mc\_i}\right)+\sum_{j=1}^{m} F_{0j}\cos\left(\alpha_{mc\_j}-\alpha\right) \\
&= F_{hyd\_in}-F_{hyd\_out}-F_{shear}-F_{frict}-F_{damp}-F_{seat}
\end{aligned}
$$

For the Needle lift greater than primary lift, the motion of **Two-spring Needle** is governed by the equation:

$$
\begin{aligned}
& X_{lift1} < \forall x \le X_{lift2} + \Delta x_{stop} : \\
& m\ddot{x} - c_{s2}\dot{x} - k_{s2}x + F_{0s2} \\
& + \sum_{i=1}^{n} c_i \left[\dot{x}\cos(\alpha - \alpha_{mc\_i}) - \dot{x}_i \cos(\alpha_{mc\_i} - \alpha_i) - \dot{y}_i \sin(\alpha_{mc\_i} - \alpha_i) \right] \cos(\alpha - \alpha_{mc\_i}) \\
& + \sum_{i=1}^{n} k_i \left[x\cos(\alpha - \alpha_{mc\_i}) - x_i \cos(\alpha_{mc\_i} - \alpha_i) - y_i \sin(\alpha_{mc\_i} - \alpha_i) \right] \cos(\alpha - \alpha_{mc\_i}) \\
& - \sum_{j=1}^{m} c_j \left[\dot{x}\cos(\alpha_{mcj} - \alpha) - \dot{x}_j \cos(\alpha_j - \alpha_{mc\_j}) - \dot{y}_j \sin(\alpha_j - \alpha_{mc\_j}) \right] \cos(\alpha_{mcj} - \alpha) \\
& - \sum_{j=1}^{m} k_j \left[x\cos(\alpha_{mcj} - \alpha) - x_j \cos(\alpha_j - \alpha_{mc\_j}) - y_j \sin(\alpha_j - \alpha_{mc\_j}) \right] \cos(\alpha_{mcj} - \alpha) \\
& - \sum_{i=1}^{n} F_{0i} \cos(\alpha - \alpha_{mc\_i}) + \sum_{j=1}^{m} F_{0j} \cos(\alpha_{mc\_j} - \alpha) \\
& = F_{hyd\_in} - F_{hyd\_out} - F_{shear} - F_{frict} - F_{damp} - F_{stop}
\end{aligned}
\tag{455}
$$

where:

m	Needle mass
x	local x-coordinate of needle
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
c_{s2} , k_{s2}	damping and stiffness constants of the secondary spring
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
F_{0s2}	preload force of the secondary spring
n, m	number of mechanical connections on input and output ends
Δx_{stop}	maximum deformation of output stop after needle-stop impact
Forces on right side of equation are defined in local x-direction:	

F_{hyd_in}	hydraulic force at input end (needle opening force) in local x-direction
F_{hyd_out}	hydraulic force at output end in local x-direction
F_{shear}	viscous friction force due to leakage (if any)
F_{frict}	Coulomb friction force
F_{damp}	damping force from squeezing fluid at needle closing
F_{seat}, F_{stop}	additional forces from needle seat (input end) and stop (output end)

All external forces on the right hand side of Equations 454 and 455 are calculated identically as for the **Standard Needle** model.

Nozzle primary spring has to be defined as an output mechanical connection from the **Needle** to a **Mechanical** or **Hydromechanical Boundary** element. The preload force of the primary spring has to be calculated from the nozzle cracking (opening) pressure.

Damping and stiffness constants of the secondary spring have to be specified directly in the Needle input dialog. If secondary spring is nonlinear, it can be also modeled as a variable mechanical connection from the **Needle** to a **Mechanical Boundary** (but not the same **Boundary** which is connected to the primary spring).

21.2.6. Equation of Motion for Elastic Needle

Motion of **Two-spring Needle** with Elastic Body in is governed by the following equations:

$$
\begin{aligned}
&\left(m_{inp}+m_{ad\_inp}\right)\ddot{x}_{inp} \\
&+\sum_{i=1}^{n} c_i\left[\dot{x}_{inp}\cos\left(\alpha-\alpha_{mc\_i}\right)-\dot{x}_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-\dot{y}_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&+\sum_{i=1}^{n} k_i\left[x_{inp}\cos\left(\alpha-\alpha_{mc\_i}\right)-x_i\cos\left(\alpha_{mc\_i}-\alpha_i\right)-y_i\sin\left(\alpha_{mc\_i}-\alpha_i\right)\right]\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&-\sum_{i=1}^{n} F_{0i}\cos\left(\alpha-\alpha_{mc\_i}\right) \\
&=+F_{hyd\_inp}-F_{C\_inp}-F_{damp}-F_{seat}+F_{stiff}+F_{damp}
\end{aligned}
$$

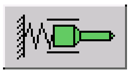
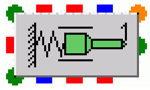
$$
\begin{aligned}
&\underline{\forall x\le X_{lift1}:} \\
&\left(m_{out}+m_{ad\_out}\right)\ddot{x}_{out} \\
&-\sum_{j=1}^{m} c_j\left[\dot{x}_{out}\cos\left(\alpha_{mcj}-\alpha\right)-\dot{x}_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+\dot{y}_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&-\sum_{j=1}^{m} k_j\left[x_{out}\cos\left(\alpha_{mcj}-\alpha\right)-x_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)+y_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&+\sum_{j=1}^{m} F_{0j}\cos\left(\alpha_{mc\_j}-\alpha\right) \\
&=F_{hyd\_out}-F_{C\_out}-F_{stop}-F_{stiff}-F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
&\underline{X_{lift1}<\forall x\le X_{lift2}+\Delta x_{stop}:} \\
&\left(m_{out}+m_{ad\_out}\right)\ddot{x}_{out}-c_{s2}\dot{x}-k_{s2}x+F_{0s2} \\
&-\sum_{j=1}^{m} c_j\left[\dot{x}_{out}\cos\left(\alpha_{mcj}-\alpha\right)-\dot{x}_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)-\dot{y}_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&-\sum_{j=1}^{m} k_j\left[x_{out}\cos\left(\alpha_{mcj}-\alpha\right)-x_j\cos\left(\alpha_j-\alpha_{mc\_j}\right)-y_j\sin\left(\alpha_j-\alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj}-\alpha\right) \\
&+\sum_{j=1}^{m} F_{0j}\cos\left(\alpha_{mc\_j}-\alpha\right) \\
&=F_{hyd\_out}-F_{C\_out}-F_{stop}-F_{stiff}-F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
F_{stiff}&=k_{ps}\left(x_{inp}-x_{out}\right), \\
F_{damp}&=c_{ps}\left(\dot{x}_{inp}-\dot{x}_{out}\right)
\end{aligned}
\qquad (456)
$$

Masses m_{inp} and m_{out} are calculated from the position of needle gravity centre (m_{inp}+m_{out}=m).

21.3. Obsolete Needle

Element Name:	**Obsolete Needle**	
Element Icon:		
Definition:	This element defines the properties an obsolete Needle model of a Standard (single-spring) Nozzle. The element is hidden (not visible in the element tree) as it should not be used with the new models.	
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 3 wire pins: 1	

> **Note:** Currently the Variable-step solver does not support this element.

Note: **Needle** element may have 2 hydraulic and up to 5 mechanical connections (in x direction). These connections are irreversible and have to be specified according to the following rules:

- **Needle** must have an **input** hydraulic connection from the **Nozzle Volume** (high pressure volume under needle guide). **Output** hydraulic connection to the outlet (spring) **Volume** or **Pressure** boundary is optional.
- **Needle** spring must be defined as an output mechanical connection to a **Mechanical** boundary (nozzle holder) or **Hydromechanical** boundary.
- In case of **Hydromechnical** boundary, mechanical connection is also a hydraulic connection, i.e. it transmits boundary pressure (if any) to **Needle**.

Note: **Obsolete Needle** model is based on the assumption that at opening needle the nozzle volume pressure is immediately acting under the area:

- for **SAC Nozzle** – from needle seat down to the nozzle sac or needle tip (depending on the needle lift). Refer to Chapters 20.1 and 20.3.
- for **VCO Nozzle** – from needle seat down to nozzle cross-section at spray holes. Refer to Chapters 20.2 and 20.4.

Note: **Obsolete Needle** model works well with the conventional injection systems (e.g. in-line pump) but is **not applicable for the common rail** systems with permanent high pressure under the needle guide.

Note: For creating a complete nozzle model, **Needle** must be connected to one of the **Nozzle Orifice** elements by a special connection.

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **Needle** may have only one hydraulic connection on each end (input and output)

☞

Note: Mechanical connections can be defined only in local x-direction.

Note: For **Needle** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system, too.

Note: Local x-axis of **Needle** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter: **Direction of Element motion**.

Needle of a **Standard SAC Nozzle** is shown in Figure 200.

21.3.1. Input Parameters

Description of input data for the **Obsolete Needle**:

Element name	The name of the element is specified as default.
Rigid Body	**Rigid Needle** is a rigid body with 1 DOF, i.e. without elasticity between the input and output end. Coordinates and velocities on input and output end of **Needle** element are the same.
Moving mass	The moving mass *m* consists of the needle mass, spring plate mass and 33% mass of the connected springs (refer to Figure 200).
Coulomb friction force	Specify constant dry friction force (if any).
Elastic Body	**Elastic Needle** implies an elastic body with the axial elasticity between the input and output end. Stiffness and damping characteristics of **Elastic Needle** are calculated according to the geometric and material data.
Geometric and Material Data …	Press this bar to open the input dialog of the geometric and material characteristics of the elastic needle.
Maximum lift of needle	Specify maximum lift of needle X_{lift} (refer to Figure 200). **Note:** If Needle has no mechanical stop, this button has to be switched off. In this case Needle stop data is automatically disabled.
NEEDLE GUIDE/SEAT	
Diameter of needle guide	Specify diameter of the needle guide d_{guide} (refer to Figure 200).
Diameter of needle seat	Specify diameter of the needle seat d_{seat} (refer to Figure 200).
HOLLOW NEEDLE	
Diameter of upper guide bore	Specify diameter of the upper bore of the hollow needle guide d_{upbore}. **Note:** If Needle is connected to RSN Throttle element (by special connection), the "Hollow Needle" option is automatically disabled.

SQUEEZING FLUID AT NEEDLE CLOSING	
Damping coefficient	Specify coefficient of velocity-proportional damping for needle closing (viscous damping of squeezing fluid). If it is not specified or set to 0, 'dry' needle motion is considered. If no a-priori estimates are available, this coefficient should be left 0.
NEEDLE SEAT/STOP	
Needle seat stiffness	Specify the needle seat stiffness on input end (k_{in_st}). Stiffness is active when needle lift $x < 0$.
Needle stop stiffness	Specify the needle stop stiffness on output end (k_{out_st}). Stiffness is active when needle lift $x > X_{lift}$.
Needle seat damping	Specify the needle seat damping on input end (c_{in_st}). Damping is active when needle lift $x < 0$ (and for pseudo-linear stop model when $\dot{x} \geq 0$).
Needle stop damping	Specify the needle stop damping on output end (c_{out_st}). Damping is active when needle lift $x > X_{lif}$ (and for pseudo-linear stop model when $\dot{x} \leq 0$).
(Nonlinear Stop model)	
Needle seat linear stiffness	Specify the needle seat linear stiffness on input end (k_{in_st}). Stiffness is active when needle lift $x < 0$.
Needle stop linear stiffness	Specify the needle stop linear stiffness on output end (k_{out_st}). Stiffness is active when needle lift $x > X_{lift}$.
Needle seat Stiffness factor	Specify the needle seat stiffness factor on input end (n_{in_st}).
Needle stop Stiffness factor	Specify the needle stop stiffness factor on output end (n_{out_st}).
Needle seat Elastic impact coef.	Specify the needle seat elastic impact coefficient on input end (α_{in_st}).
Needle stop Elastic impact coef.	Specify the needle stop elastic impact coefficient on output end (α_{out_st}).
SEAT/STOP MODEL	
Linear Seat/Stop model	This is a simple, purely linear contact model where stiffness and damping are constant and contact force is zero whenever the distance is positive (see equations **64** and **67**). (refer to **Figure 54**).
Pseudo-linear Seat/Stop model	This model is derived from the linear model by introducing additional switching function (see equations **65** and **68**). The contact model describes contact in a way that avoids both negative contact force and the discontinuities at the beginning and end of contact (refer to **Figure 55**).

Non-linear Seat/Stop model	Linear model, commonly used to describe the interaction with stiff environments, sometimes cannot be applied to soft materials, where viscous effects are relevant. For the better physical consistency in describing this behavior a non-linear Hunt-Crossley model[40] model is used.
DIRECTION OF ELEMENT MOTION	
x(e1)	Specify the unit vector of element x-axis in the global coordinate system. $\vec{X}_{loc} = e_1\vec{X}_{gl} + e_2\vec{Y}_{gl} + e_3\vec{Z}_{gl}$,
y(e2)	where $\vec{X}_{gl}, \vec{Y}_{gl}, \vec{Z}_{gl}$ are unit vectors in the global coordinate system, e_1, e_2, e_3 are the unit vector components in the global *x*, *y* and *z* direction.
z(e3)	Default values for unit vector are 1. / 0. / 0. (global and local coordinate systems are identical).

21.3.2. Initial Conditions

Initial Conditions of Obsolete Needle are identical to the **Initial Conditions of Standard Piston** (refer to Section **7.1.3**).

21.3.3. Modifiable Parameters

Modifiable Parameters of **Obsolete Needle** are the same as modifiable parameters of **Standard Needle** described in Section 21.4.3.

Path: **Element** | **Modify**

To activate modification, the option button to the left of modifiable parameter name must be pressed. When modification is activated, Modification Table must be defined (refer to Section **26.2.3.2**). Open by pressing **Modify...** bar.

Modifiable parameters:

Coulomb friction force	SI Unit:	N
squeezing fluid damping at needle closing	SI Unit:	Ns/m
maximum lift of needle	SI Unit:	M

[40] K. H. Hunt, F. R. E. Crossley, *Coefficient of Restitution Ineterpreted as Damping in Vibroimpact*, ACME Journal of Applied Mechanics, 1975.

21.3.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

x-coordinate of needle (input end)	Local x-coordinate (lift) of input end of Needle.
needle velocity in x-direction (input end)	Velocity in local x-direction of input end of Needle.
needle acceleration in x-direction (input end)	Acceleration in local x-direction of input end of Needle.
x-coordinate of needle (output end)	Local x-coordinate (lift) of output end of Needle.
needle velocity in x-direction (output end)	Velocity in local x-direction of output end of Needle.
needle acceleration in x-direction (output end)	Acceleration in local x-direction of output end of Needle.
Needle lift, velocity and acceleration are calculated according to the dynamic equation of motion (Equation 457) discussed in Section 21.3.5.	
hydraulic force acting on needle	Sum of hydraulic forces acting on the input and output end of Needle (input force minus output force).
resultant force	Resultant force of needle is the sum of all connection forces and external forces acting on it. $F_R = m\ddot{x}$
mechanical force	Mechanical force is the sum of the preload forces, stiffness and damping forces, Coulomb friction force and forces from the needle stops.
shear force from leakage	Shear force is calculated in **Annular gap** element, which is connected to needle element via special connection.

The output parameters of the **Obsolete Needle** are identical to the output parameters dialog of the **Standard Needle** except that there are no outputs in y and z directions (**Obsolete Needle** is only a 1DOF body). Note that hydraulic (opening) force for **Standard** and **Obsolete Needle** is calculated differently (refer to Section 0).

21.3.5. Equation of Motion for Rigid Needle

Motion of **Rigid Needle** in local coordinate system is governed by the following equation:

$$\begin{aligned}
& m\ddot{x} \\
& +\sum_{i=1}^{n} c_i \left[\dot{x}\cos(\alpha-\alpha_{mc_i}) - \dot{x}_i \cos(\alpha_{mc_i}-\alpha_i) - \dot{y}_i \sin(\alpha_{mc_i}-\alpha_i)\right]\cos(\alpha-\alpha_{mc_i}) \\
& +\sum_{i=1}^{n} k_i \left[x\cos(\alpha-\alpha_{mc_i}) - x_i \cos(\alpha_{mc_i}-\alpha_i) - y_i \sin(\alpha_{mc_i}-\alpha_i)\right]\cos(\alpha-\alpha_{mc_i}) \\
& -\sum_{j=1}^{m} c_j \left[\dot{x}\cos(\alpha_{mcj}-\alpha) - \dot{x}_j \cos(\alpha_j-\alpha_{mc_j}) + \dot{y}_j \sin(\alpha_j-\alpha_{mc_j})\right]\cos(\alpha_{mcj}-\alpha), \qquad (457) \\
& -\sum_{j=1}^{m} k_j \left[x\cos(\alpha_{mcj}-\alpha) - x_j \cos(\alpha_j-\alpha_{mc_j}) + y_j \sin(\alpha_j-\alpha_{mc_j})\right]\cos(\alpha_{mcj}-\alpha) \\
& -\sum_{i=1}^{n} F_{0i}\cos(\alpha-\alpha_{mc_i}) + \sum_{j=1}^{m} F_{0j}\cos(\alpha_{mc_j}-\alpha) \\
& = F_{hyd_in} - F_{hyd_out} - F_{shear} - F_{frict} - F_{damp} - F_{seat} - F_{stop}
\end{aligned}$$

where:

m	Needle mass
x	local x-coordinate of needle
x_i	local x-coordinate of i-th connected element on input end
y_i	local y-coordinate of i-th connected element on input end
x_j	local x-coordinate of the j-th connected element on output end
y_j	local y-coordinate of j-th connected element on output end
α	angle between element motion of direction (x-coordinate) and global x-coordinate
α_i	angle between x-coordinate of i-th connected element and global x-coordinate
α_j	angle between x-coordinate of j-th connected element and global x-coordinate
α_{mc_i}	angle between i-th mechanical connection and global x-coordinate
α_{mc_j}	angle between j-th mechanical connection and global x-coordinate
c_i ,k_i	damping and stiffness coefficients (functions) of the i-th mechanical connection on input end
c_j , k_j	damping and stiffness coefficients (functions) of the j-th mechanical connection on output end
F_{0i} ,F_{0j}	preload forces of the i-th input and j-th output mechanical connections, respectively
n, m	number of mechanical connections on input and output ends
Forces on right side of equation are defined in local x-direction:	
F_{hyd_in}	hydraulic force at input end (needle opening force) in local x-direction

F_{hyd_out}	hydraulic force at output end in local x-direction
F_{shear}	viscous friction force due to leakage (if any)
F_{frict}	Coulomb friction force
F_{damp}	damping force from squeezing fluid at needle closing
F_{seat}, F_{stop}	additional forces from needle seat (input end) and stop (output end)

For all **Needle** types excluding the **RSN Needle**, the opening force F_{hyd_in} consists of two components:

$$F_{hyd\_in} = F_{guide} + F_{tip}, \tag{458}$$

where F_{guide} is the hydraulic force acting upon needle from the nozzle volume (under needle guide) and F_{tip} is the hydraulic force acting on the needle tip.

Depending on the needle position, the guide force F_{guide} is calculated from the equation:

$$\begin{aligned} \forall\, x \le 0:\ F_{guide} &= p_{in}\left(A_{guide} - A_{seat}\right) = \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{seat}^2\right), \\ \forall\, x > 0:\ F_{guide} &= p_{in}\left(A_{guide} - A_{tip}\right) = \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{tip}^2(x)\right), \end{aligned} \tag{459}$$

where p_{in} is the input pressure (in nozzle volume), A_{guide}, A_{seat} and A_{tip} are the cross-sectional areas of needle guide, needle seat and tip, respectively, d_{guide} and d_{seat} are the diameters of needle guide and seat and $d_{tip}(x)$ is the lift-dependent diameter of needle tip (subjected to the sac pressure).

Comparison of Equations 414 and 459 leads to the following conclusion: at the instant of needle opening, the guide force of **Standard Needle** model is significantly smaller than the guide force of **Obsolete Needle** model because dseat > dtip(x).

For **Hollow Needle**, the guide force F_{guide} is given by:

$$\begin{aligned} \forall\, x \le 0:\ F_{guide} &= \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{upbore}^2 - d_{seat}^2\right), \\ \forall\, x > 0:\ F_{guide} &= \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{upbore}^2 - d_{tip}^2(x)\right), \end{aligned} \tag{460}$$

where d_{guide}, d_{upbore} and d_{seat} are the diameters of the lower needle guide, upper guide bore and needle seat, respectively.

For **RSN Needle** with a collar (connected by a special connection to **RSN Throttle** element), needle opening force F_{hyd_in} consists of three components:

$$F_{hyd\_in} = F_{guide} + F_{cl} + F_{tip}, \tag{461}$$

where F_{cl} is the hydraulic force acting on the needle collar (refer to Chapter 20.5.5). Guide force F_{guide} is calculated from the equation:

$$F_{guide} = p_{in}\left(A_{guide} - A_{cl}\right) = \frac{\pi}{4} p_{in}\left(d_{guide}^2 - d_{cl}^2\right), \tag{462}$$

where A_{cl} is the cross-sectional area and d_{cl} is the diameter of the needle collar defined in the input dialog of **RSN Throttle**.

Area A_{tip} and diameter $d_{tip}(x)$ are calculated for both **Basic** and **Extended SAC** and **VCO Nozzles** (refer to Chapter 20). For the **SAC Nozzles**, A_{tip} and $d_{tip}(x)$ generally depend on the needle lift as shown below:

$$\begin{aligned} &\forall\, d_{sac} > \left(d_{tip0} + x\sin\alpha_{seat}\right):\ d_{tip}(x) = d_{sac} - x\sin\alpha_{seat}, \\ &\forall\, d_{sac} \le \left(d_{tip0} + x\sin\alpha_{seat}\right):\ d_{tip}(x) = d_{tip0}\ , \end{aligned} \tag{463}$$

where d_{sac} and d_{tip0} are the input diameters of nozzle sac and needle tip, respectively, and α_{seat} is the angle of needle seat.

For **Basic** and **Extended VCO Nozzles**, $d_{tip}(x) = d_{nsh} = const.$, where d_{nsh} is the nozzle diameter at the cross-section of spray holes.

For **Basic** and **Extended SAC Nozzles**, needle tip force F_{tip} is calculated from the formula:

$$F_{tip} = p_{sac}A_{tip} = \frac{\pi}{4} p_{sac} d_{tip}^2(x), \tag{464}$$

where p_{sac} is the pressure in the nozzle sac.

For **Basic** and **Extended VCO Nozzles**, needle tip force F_{tip} is calculated from:

$$F_{tip} = p_{nsh}A_{nsh} = \frac{\pi}{4} p_{nsh} d_{nsh}^2\ , \tag{465}$$

where p_{nsh} is the pressure under the area of nozzle spray holes.

All other forces on the right hand side of Equation 457 are calculated identically as for **Standard Needle** model.

Remember that **Nozzle** spring must be defined as an output mechanical connection from the **Needle** to a **Mechanical** or **Hydromechanical Boundary** element. The preload force of nozzle spring has to be calculated from the nozzle cracking (opening) pressure.

21.3.6. Equation of Motion for Elastic Needle

Motion of **Needle with Elastic Body** is governed by the following equations:

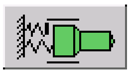
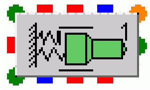
$$
\begin{aligned}
&\left(m_{inp} + m_{ad\_inp}\right)\ddot{x}_{inp} \\
&+ \sum_{i=1}^{n} c_i \left[\dot{x}_{inp}\cos\left(\alpha - \alpha_{mc\_i}\right) - \dot{x}_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - \dot{y}_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
&+ \sum_{i=1}^{n} k_i \left[x_{inp}\cos\left(\alpha - \alpha_{mc\_i}\right) - x_i \cos\left(\alpha_{mc\_i} - \alpha_i\right) - y_i \sin\left(\alpha_{mc\_i} - \alpha_i\right)\right]\cos\left(\alpha - \alpha_{mc\_i}\right) \\
&- \sum_{i=1}^{n} F_{0i}\cos\left(\alpha - \alpha_{mc\_i}\right) \\
&= +F_{hyd\_inp} - F_{C\_inp} - F_{damp} - F_{seat} + F_{stiff} + F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
&\left(m_{out} + m_{ad\_out}\right)\ddot{x}_{out} \\
&- \sum_{j=1}^{m} c_j \left[\dot{x}_{out}\cos\left(\alpha_{mcj} - \alpha\right) - \dot{x}_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + \dot{y}_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
&- \sum_{j=1}^{m} k_j \left[x_{out}\cos\left(\alpha_{mcj} - \alpha\right) - x_j \cos\left(\alpha_j - \alpha_{mc\_j}\right) + y_j \sin\left(\alpha_j - \alpha_{mc\_j}\right)\right]\cos\left(\alpha_{mcj} - \alpha\right) \\
&+ \sum_{j=1}^{m} F_{0j}\cos\left(\alpha_{mc\_j} - \alpha\right) \\
&= F_{hyd\_out} - F_{C\_out} - F_{stop} - F_{stiff} - F_{damp}
\end{aligned}
$$

$$
\begin{aligned}
F_{stiff} &= k_{ps}\left(x_{inp} - x_{out}\right), \\
F_{damp} &= c_{ps}\left(\dot{x}_{inp} - \dot{x}_{out}\right)
\end{aligned}
$$

Masses m_{inp} and m_{out} are calculated from the position of needle gravity centre (m_{inp}+m_{out}=m).

21.4. Two-spring Needle (obsolete)

Element Name:	**Two-Spring Needle (obsolete model)**	
Element Icon:		
Definition:	This element defines the properties an obsolete Needle model of a Two-spring injector (TSI) Nozzle. The element is hidden (not visible in the element tree) as it should not be used with the new models.	
Connecting pins:	standard pins: 7 (5 mechanical and 2 hydraulic) special pins: 3 wire pins: 1	

Note: Currently the Variable-step solver does not support this element.

Note: **Needle** element may have 2 hydraulic and up to 5 mechanical connections (in x direction). These connections are irreversible and have to be specified according to the following rules:

- **Needle** must have an **input** hydraulic connection from the **Nozzle Volume** (high pressure volume under needle guide). **Output** hydraulic connection to the outlet (spring) **Volume** or **Pressure** boundary is optional.
- **Needle primary spring** must be defined as an output mechanical connection to a **Mechanical** (nozzle holder) or **Hydromechanical** boundary (nozzle holder plus outlet pressure). **Secondary spring** data have to be defined directly in the input dialog.
- In case of **Hydromechnical** boundary, mechanical connection is also a hydraulic connection, i.e. it transmits boundary pressure (if any) to **Needle**.

Note: **Obsolete Needle** model is based on the assumption that at opening needle the nozzle volume pressure is immediately acting under the area:

- for **SAC Nozzle** – from needle seat down to the nozzle sac or needle tip (depending on the needle lift).

Note: **Obsolete Needle** model works well with the conventional injection systems (e.g. in-line pump) but is **not applicable for the common rail** systems with permanent high pressure under the needle guide.

Note: For creating a complete nozzle model, **Needle** must be connected to one of the **Nozzle Orifice** elements by a special connection.

Note: Hydraulic connections can be defined only in local x-direction. Multiple connections to the same element are not allowed (all connections except the first will be ignored). **Needle** may have only one hydraulic connection on each end (input and output)

☞

Note: Mechanical connections can be defined only in local x-direction.

Note: For **Needle** element, input parameters and initial conditions have to be defined in local x coordinate system. Output parameters as coordinates, velocities and accelerations are calculated in local coordinate system, too.

Note: Local x-axis of **Needle** may be **translated** and **rotated** in global coordinate system. Rotation is defined through parameter: **Direction of Element motion**.

Needle of a **Two-Spring Injector Nozzle** with primary and secondary springs is shown in Figure 225.

21.4.1. Input Parameters

The input data dialog box of **Two-spring Needle (obsolete)** is identical to the input data dialog box of standard **Two-spring Needle** except for the title and option for pressure distribution along needle seat. These are explained in detail in Chapter 21.2.

21.4.2. Initial Conditions

The **Initial Conditions** of the **Two-spring Needle (obsolete)** are identical to the **Initial Conditions** of the **Obsolete Needle**. Refer to Section 21.3.2 for more information.

21.4.3. Modifiable Parameters

Modifiable Parameters of **Two-spring Needle (obsolete)** are the same as modifiable parameters of standard **Two-spring Needle** described in Section 21.4.3.

21.4.4. Output Parameters

Output Parameters of **Two-spring Needle (obsolete)** are identical to the **Output Parameters** of standard **Two-spring Needle** shown in Section 21.4.4. However, hydraulic (opening) force is calculated differently (refer to Section 21.2.5).

21.4.5. Equation of Motion

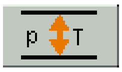
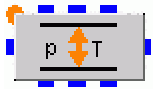
Motion of **Two-spring Needle (obsolete)** is governed by identical differential equations as the motion of standard **Two-spring Needle.**

Needle guide and tip forces are calculated from the corresponding equations of **Obsolete Needle** model (refer to Section 21.3.5). Calculation of other external forces is identical to the previous **Needle** models.

Remember that **Nozzle** primary spring must be defined as an output mechanical connection from the **Needle** to a **Mechanical** or **Hydromechanical Boundary** element. Damping and stiffness constants of the secondary spring have to be specified directly in the **Needle** input dialog. If secondary spring is nonlinear, it can be also modeled as a variable mechanical connection from the **Needle** to a **Mechanical Boundary** (but not the same **Boundary** which is connected to the primary spring).

22. GAS

22.1. Gas Pressure/Temperature Boundary

Element Name:	**Gas Pressure/Temperature Boundary**	
Element Icon:		
Definition:	This element serves to define the gas pressure and temperature on the outer connections (boundaries) of the system as functions of time or reference angle.	
Connecting pins:	standard pins: 8 (all hydraulic) special pins: 0 wire pins: 1 Pressure/temperature is the same at all pins.	

Note: *x-direction* is the only hydraulic direction.

22.1.1. Input Parameters

Description of input data:

Element name	The name of the element is specified as default.
Shift Time/Angle relative to calculation start	Specify the Shift time/angle for the first table column. It will be added to the time/angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Copy data for each cycle (360 deg Ref. angle):	Activate this button to specify of the values in table as a function of time (or reference angle) only for one cycle (360 degrees of Reference Angle). If more than one revolution of the cam is considered in the calculation, the program will copy data from the table for each new cycle.
Pressure and Temperature vs. Time/Reference Angle:	Specify the Pressure and/or Temperature as a function of time (or reference angle) in form of table.

LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **8.1.8**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored into GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)

22.1.2. Initial Conditions

No initial conditions can be specified for Gas Pressure/Temperature Boundary element.

22.1.3. Modify Parameter

Gas Pressure/Temperature Boundary has no modifiable parameters.

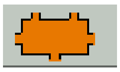
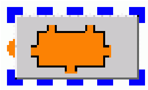
22.1.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box on the left of the parameter name has to be checked.

Pressure	Actual pressure in **Gas Pressure/Temperature Boundary**.
Temperature	Actual temperature in **Gas Pressure/Temperature Boundary**.
For **Gas Pressure/Temperature Boundary** element, the input and output data is the same.	

22.2. Gas Volume

Element Name:	**Gas Volume**	
Element Icon:		
Definition:	This element serves to define the properties of a Gas Volume.	
Connecting pins:	standard pins: 10 (all hydraulic) special pins: 0 wire pins: 1	

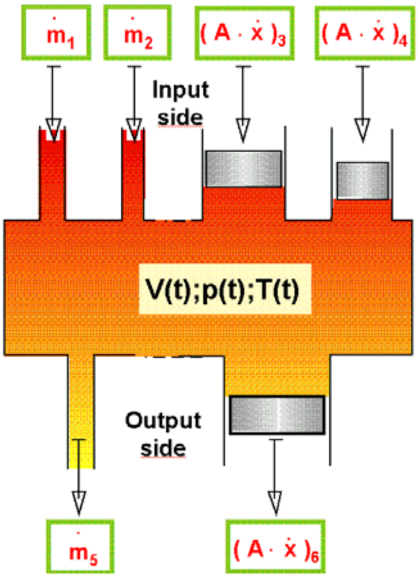


Figure 226: Gas Volume

☞	**Note:** If the **Gas Volume** element is only attached to the **Piston** element, it can be thought of as isentropic (refer to Section 22.2.5.1). The pressure and volume of the gas trapped are calculated assuming internally reversible adiabatic process. The entropy of fixed mass (no mass inflow and mass outflow) remains constant. This is sufficient to simulate the compression and expansion processes taking place in hydropneumatic accumulators (see Figure 227) that are governed in this program by the ideal gas laws.

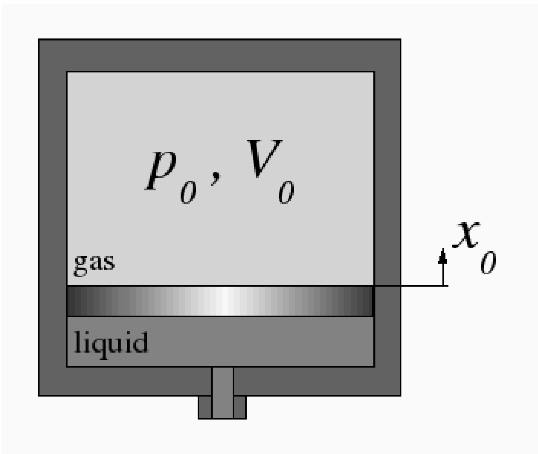


Figure 227: Hydropneumatic Accumulator

22.2.1. Input Parameters

Description of input data for the **Gas Volume**:

Element name:	The name of the element is specified as default.
Gas Properties	Press this bar to specify the local gas properties (refer to Section 22.2.6).

Initial volume	Specify volume at start of calculation. If a piston-type element is attached to **Volume,** initial volume will change according to the position of the piston. Volume and pressure will vary.
LINK TO EXCITE POWER UNIT	
Element reaction force on housing is calculated (refer to section **8.1.8**).	
Element (separate force)	The calculated force is stored in a GIDas file. (<Hydsim_Model>_ENF_<Element_Name>_<Element_ID>.GID)
Macroelement (resultant force)	The calculated force with all other macro element forces is summarized and results are stored in a GIDas file. (<Hydsim_Model>_ENF_<Macroelement_Name>.GID)

22.2.2. Initial Condition

Path: **Element**| **Initial Conditions**

Description of initial conditions:

Pressure:	Specify initial pressure in the **Volume** at the beginning of the calculation.
Temperature:	Specify initial temperature in **Volume** at the beginning of the calculation.

☞ **Note:** By default initial pressure and temperature are set to 1 bar and 20 deg C, respectively.

22.2.3. Modifiable Parameters

Gas Volume has no modifiable parameters.

22.2.4. Output Parameters

Path: **Element**| **Store Results**

To activate output parameter, the option box in front of output parameter must be activated.

Description of output parameters:

actual volume	Actual volume is calculated from the following equation: $$V(x) = V_0 - \sum_{i=1}^{n_2} A_i x_i + \sum_{j=1}^{l_2} A_j x_j$$, where V_0 is the initial volume and x_i and x_j are the displacements of piston-type elements connected to **Volume** by *i-th* input and *j-th* output connection, respectively.
gas pressure	Gas pressure is calculated from Equation 471 or for isentropic case 473.

max. pressure till selected point	Maximum volume pressure from the calculation start will be saved.
gas temperature	Gas temperature is calculated from Equation 470 or for isentropic case 474.
max. temperature till selected point	Maximum volume temperature from the calculation start will be saved.
gas density	Gas density is calculated from Equation 475.
specific enthalpy	For the perfect gas, the specific enthalpy is a pure temperature function. It is calculated by: $$h_i = \int_{T_0}^{T_i} c_p(T)dT ,$$ where cp is the specific heat of gas at constant pressure.
actual mass	Actual mass is calculated from the following equation: $$m(t) = m_0 + \int_0^t \dot{m}(t)dt ,$$ $$\dot{m}(t) = \sum_{i=1}^{n1} \dot{m}_i - \sum_{j=1}^{l1} \dot{m}_j ,$$ where m_0 is the initial mass and $\dot{m}_i$ and $\dot{m}_j$ are the mass inflows and outflows through gas orifice-type elements connected to **Gas Volume** by *i-th* input and *j-th* output connection, respectively.
mass rate at volume output end /time	Sum of all mass rates at input end in **Time** domain.
mass rate at volume output end /angle	Sum of all mass rates at input end in **Reference angle** domain or **Crank angle** domain. **Note:** Output angle domain is defined in **Calculation Control** (refer to Chapter 26.2.6.3). **Note:** Output values will be 0, if the **Time** domain button is active in **Calculation Control**.
mass rate at volume output end /time	Sum of all mass rates at output end in **Time** domain.
mass rate at volume output end /angle	Sum of all mass rates at output end in **Reference angle** domain or **Crank angle** domain.

22.2.5. Governing Equations

Temperature and pressure in **Gas Volume** are calculated from the continuity equation, conservation of energy and equation of state for ideal gases.

Continuity equation (conservation of mass) is given by:

$$\sum \dot{m} = \sum \left(\frac{dm}{dt} \right)_i , \qquad (466)$$

where:

$\dot{m}$	rate of mass change in Volume
i	index for inflows and outflows (inflows have positive sign and outflows – negative sign)

Equation of energy conservation follows from the first law of thermodynamics:

$$\frac{dU}{dt} = \frac{dH_{in}}{dt} - \frac{dH_{out}}{dt} + \frac{dW}{dt} , \qquad (467)$$

where:

$\frac{dU}{dt}$	rate of change of internal energy of fluid
$\frac{dH_{in}}{dt}, \frac{dH_{out}}{dt}$	enthalpy of inflows and outflows, respectively
$\frac{dW}{dt}$	rate of work done on the fluid in Volume

Energy conservation equation can be expressed in the simplified form:

$$\frac{d(mu)}{dt} = \sum_i \dot{m}_i c_{pi} T_i - p \frac{dV}{dt} \qquad (468)$$

or

$$\frac{d(mc_v T)}{dt} = \sum_i \dot{m}_i c_{pi} T_i - p \frac{dV}{dt} \qquad (469)$$

where:

m	mass in the volume
$\dot{m}$	rate of gas mass change in volume ($= \sum \dot{m}_i$)
c_v	specific heat capacity of gas at constant volume
c_{pi}	specific heat capacity at constant pressure of *i-th* inflow / outflow
T_i	temperature of *i-th* inflow / outflow
T	gas temperature in the volume

p	gas pressure in the volume
$\frac{dV}{dt}$	rate of Volume change due to compression/expansion

From the above equation we obtain temperature time derivative in the form:

$$\frac{dT}{dt} = \frac{1}{mc_v}\left(\sum_i \dot{m}_i c_{pi} T_i - c_v T \sum_i \dot{m}_i - p\frac{dV}{dt}\right) \tag{470}$$

Pressure is calculated from equation of state for ideal gas:

$$pV = mRT \tag{471}$$

From it pressure in gas volume in given by:

$$p = \frac{mRT}{V} \tag{472}$$

22.2.5.1. Isentropic Process

The Isentropic process involves no heat transfer (adiabatic) and no irreversibilities within the system (internally reversible). The entropy remains constant.

Based of the perfect gas law, the isentropic process is given by:

$$pv^{\kappa} = const., \tag{473}$$

where v is the specific volume and κ is specific heat ratio of gas.

Temperature is calculated from the equation:

$$Tv^{\kappa-1} = const.. \tag{474}$$

Gas density is calculated from the state equation:

$$\rho = \frac{p}{RT}. \tag{475}$$

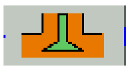
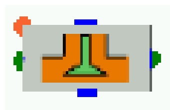
22.2.6. Local Gas Properties

There are three options within the **Local Gas Properties** dialog:

1. **Global gas properties** [default].
   Gas properties are taken from the table of **Global Gas Properties** (refer to 24.2.5.5 for more information).

2. **Gas properties from Property Database**
   Gas properties are chosen from the menu in Property Database (refer to Chapter 24.2.5.8 for more information)
   - Gas name [name of specie]
3. **Variable gas properties**
   The tabular values of gas properties as a function of temperature have to be specified for the following properties:
   - Specific heat c_p [Gas Constant Units]
   - Specific heat ratio κ [Ratio Units].

22.3. Lift-controlled Throttle

Element Name:	**Lift-controlled Throttle**	
Element Icon:		
Definition:	This element serves to define a gas throttle controlled by a position of one or two mechanical bodies. Flow area is a function of absolute or relative lift.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 2 wire pins: 1 Only one input and one output hydraulic connection can be specified.	

Note: **Lift-controlled Throttle** alone is not a physical element but basically a control function. Hence, it needs a link to a mechanical body (piston-type element).
Mechanical body has to be connected to **Lift-controlled Throttle** by special connection (green line). With this connection, **Lift-controlled Throttle** will receive information about the position of the body.

Note: If **Lift-controlled Throttle** is connected via two special connections to two mechanical bodies (piston-type elements), its flow area is a function of the coordinate difference (relative lift) between these two elements. In this case the mechanical bodies must be interconnected via mechanical connection.

22.3.1. Input Parameters

Description of input data for the **Lift-controlled Throttle**:

Element name	The name of the element is specified as default.
Gas Properties	Press to specify the local gas properties (refer to Section 22.2.6).
EFFECTIVE CROSS-SECTIONAL FLOW AREA (my*A)	
Factor for 2nd column (my)	This button activates the multiplication factor for 2[nd] column. Values in the second column will be multiplied with scaling factor as soon as **Run/Run Sets/Restart** is selected in the **Simulation PullDownMenu**. It is convenient to use **Scaling factor for 2nd column** as discharge coefficient μ (if it is constant and valve closing/opening areas are available in absolute values not pre-multiplied by μ.
Effective Flow Area (my*A)	Specify coordinate (lift) of mechanical body (or relative lift between two bodies) in ascending order in the first column. Specify effective cross-sectional flow areas of valve seat in the second column. Effective cross-sectional flow area of **Lift-controlled Throttle** is calculated according to the actual position of the mechanical body. General rules for the specification of table data: • The intermediate values are bridged by linear interpolation. • If the actual body lift exceeds the maximum value in the table, the last specified value of flow area is used for the further calculation. • If the actual body lift is less than the lift specified in the first table row, the first specified value of flow area will be used. Refer to the description of tables in **Switch Valve** in Section 16 for entering data.

22.3.2. Initial Conditions

Initial conditions cannot be specified for **Lift-controlled Throttle**.

22.3.3. Modify Parameter

Modifiable parameters cannot be specified for **Lift-controlled Throttle**.

22.3.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

mass flow rate through valve	Mass rate through **Lift-controlled Throttle** is calculated from Equation 476 (refer to Section 22.3.5). Volumetric flow rate through valve in **Time** domain.
mass flow rate through valve	Mass rate through valve in **Reference angle** domain or **Crank angle** domain.
cumulative mass rate through valve	Cumulative mass rate per calculation range.
effective cross-sectional valve flow area	BOOST Hydsim calculates effective cross-sectional flow area from input table according to the actual lift of mechanical body linked to **Lift-controlled Throttle** by special connection.
lift of connected body (absolute/relative)	If **Lift-controlled Throttle** is connected via only one special connection to mechanical body (piston-type element), BOOST Hydsim takes coordinate of this element. If **Lift-controlled Throttle** is connected via two special connections to two mechanical bodies (piston-type elements), BOOST Hydsim calculates coordinate difference (relative lift) between these two elements.
flow velocity through valve	Gas flow velocity in the throttle (narrowest area)
Mach number	Mach number in the throttle

22.3.5. Model of Gas Flow through Throttles

The flow model through the gas valves is based on the energy equation, continuity equation and the equation for the isentropic change of state:

$$\dot{m} = \mu A p_{upst} \psi \sqrt{\frac{2}{R_g T_{upst}}} \,, \qquad (476)$$

where $\dot{m}$ is the gas mass rate, p_{upst} and T_{upst} are pressure and temperature on upstream end, μA is effective cross-sectional flow area (at valve seat), R_g is gas constant and ψ is the function dependent on the flow type. For this, function ψ is given by the following equation:

$$\psi = \sqrt{\frac{\kappa}{\kappa - 1}\left[\left(\frac{p_{dwst}}{p_{upst}}\right)^{\frac{2}{\kappa}} - \left(\frac{p_{dwst}}{p_{upst}}\right)^{\frac{\kappa+1}{\kappa}}\right]} \,, \qquad (477)$$

where p_{dwst} is the pressure on downstream end κ is the specific heat ratio.

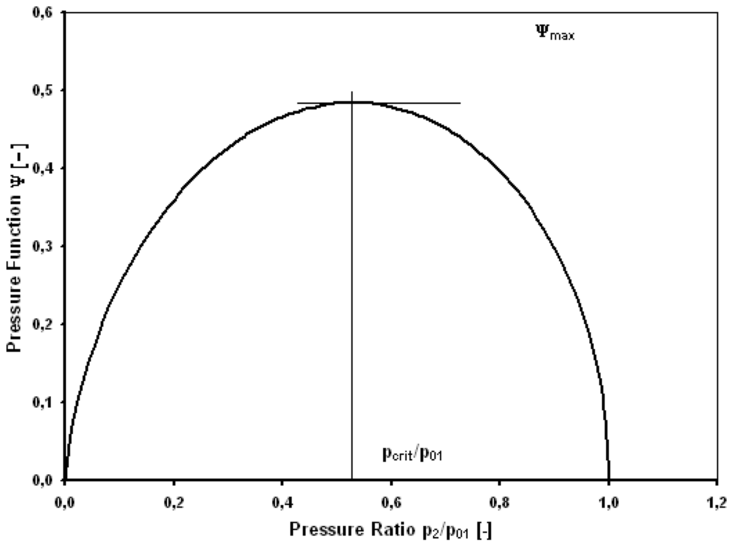


Figure 228: Pressure function vs. downstream/upstream pressure ratio

In the case of subsonic flow, the pressure ratio, which is defined as the downstream static pressure divided by the upstream stagnation pressure, is higher than the critical pressure ratio and less than or equal to 1. The pressure function ψ follows the curve shown in Figure 228 for the given range of pressure ratios.

If the pressure ratio drops to the critical value

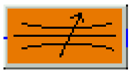
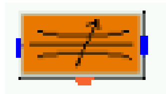
$$\frac{p_{crit}}{p_{upst}} = \left(\frac{2}{\kappa+1}\right)^{\frac{\kappa}{\kappa-1}}, \tag{478}$$

flow in the throttle becomes sonic (Mach number 1). The pressure function ψ reaches its maximum at the critical pressure ratio. The actual value of $\psi_{\max}$ depends on the pressure ratio:

$$\psi = \psi_{\max} = \left(\frac{2}{\kappa+1}\right)^{\frac{1}{\kappa-1}} \cdot \sqrt{\frac{\kappa}{\kappa+1}}, \tag{479}$$

Values of the pressure function ψ shown in Figure 228, for pressure ratios less than the critical pressure ratio are valid only for supersonic flow in the throttle. However, supersonic flow can never be achieved just by lowering the back pressure, but requires a special shape of the pipe upstream of the orifice (Laval-Nozzle). In BOOST Hydsim gas flow elements this geometry is not defined, therefore gas flow is limited to sonic flow regime (Mach=1).

22.4. Flow Area vs. Time/CA

Element Name:	**Flow Area vs. Time/CA**	
Element Icon:		
Definition:	This element serves to define a gas throttle with flow area as a function of time or crank angle.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 1 Only one input and one output hydraulic connection can be specified.	

22.4.1. Input Parameters

Description of input data for the **Flow Area vs. Time/CA element**:

Element name	The name of the element is specified as default.
Gas Properties	Press this bar has to specify the local gas properties (refer to Section 8.4.6).

EFFECTIVE FLOW AREA (my*A)	
Shift Time/CA relative to calculation start	Specify the Shift time/crank angle for the first table column. It will be added to the time/crank angle values in the table at calculation start.
Scaling factor for 1st column:	This button activates scale factor for 1st column in the table. Values in first column will be multiplied with scale factor as soon as calculation is started.
Factor for 2nd column (my)	This button activates the multiplication factor for 2 nd column. For alternative button refer to **Flow discharge coef. (** μ **)** Values in second column will be multiplied with scaling factor as soon as **Run/Run Sets/Restart** is selected in the **Simulation PullDownMenu**. It is convenient to use **Scaling factor for 2nd column** as discharge coefficient μ (if it is constant and valve closing/opening areas are available in absolute values not pre-multiplied by μ .
Copy data for each cycle (720 deg Crank angle):	Activate this button to specify the values in the table as a function of time (or crank angle) only for one cycle (720 degrees of Crank Angle). If more than two revolutions of the crankshaft are considered in the calculation, the program will copy data from the table for each new cycle.

	Note: If the Copy data button is **active** in global dialog Simulation/Boundary Data for each cycle, then it will have dominant status and it will get active at Simulation/Run, Run Sets or Restart.
Effective Flow Area (my*A)	Specify time/crank angle in ascending order in first column. Specify effective cross-sectional flow areas of valve seat in second column. Effective cross-sectional flow area of **Flow Area vs. Time/CA** element is calculated according to the actual time/crank angle. General rules for the specification of table data: • The intermediate values are bridged by linear interpolation. • If the actual time/CA exceeds the maximum value in the table, the last specified value of flow area is used for the further calculation. • If the actual time/CA is less than the lift specified in the first table row, the first specified value of flow area will be used. Refer to the description of tables in **Variable Throttle** in Section 16 for entering data.
FLOW DISCHARGE COEFFICIENT (my)	
Discharge coef. for forward flow	Specify the flow discharge coefficient for the forward flow μ_{in} (from input to output end).
Discharge coef. for backward flow	Specify the flow discharge coefficient for the backward flow μ_{out} (from output to input end).

☞ **Note:** **Flow discharge coef. (** μ **)** allows to specify different flow discharge coefficients for the forward and backward flow. If it is activated, multiplication factor for 2nd table column μA is automatically switched off.

22.4.2. Initial Conditions

Initial conditions cannot be specified for **Flow Area vs. Time/CA** element.

22.4.3. Modify Parameter

Modifiable parameters cannot be specified for **Flow Area vs. Time/CA** element.

22.4.4. Output Parameters

Path: **Element** | **Store Results**

To activate output parameter, the check box in front of output parameter has to be checked.

Description of output parameters:

mass flow rate through valve	Mass rate through **Flow Area vs. Time/CA** element is calculated from Equation 476 (refer to Section 22.3.5). Volumetric flow rate through valve in **Time** domain.
mass flow rate through valve	Mass rate through valve in **Reference angle** domain or **Crank angle** domain.
cumulative mass rate through valve	Cumulative mass rate per calculation range.
effective cross-sectional valve flow area	BOOST Hydsim calculates effective cross-sectional flow area from input table according to the actual **Time** or **Crank angle**.
flow velocity through valve	Gas flow velocity in the throttle (narrowest area)
Mach number	Mach number in the throttle

22.4.5. Flow Equation

Flow rate through **Flow Area vs. Time/CA** element is calculated by the same equation (Equation 476) as the flow rate through **Lift-controlled Throttle** (refer to Section 22.3.5). If the option **Flow discharge coef. (** μ **)** is active, then $\mu = \mu_{in}$ for the forward flow (from input to output end) and $\mu = \mu_{out}$ for the backward flow (from output to input end).

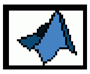
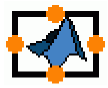
23. CONTROL

There are two main types of engine control element available in BOOST Hydsim:

- External Link Elements:
  - MATLAB® API (Simulink)
  - MATLAB® API (m-function)
  - MATLAB® DLL
- Internal Control Element:
  - PID Controller

The link to an External Control Element Library is a complementary element to the Engine Control Unit (ECU) element. It may be used to incorporate complex models of engine control and management systems developed with MATLAB/SIMULINK (MATLAB-API Element, MATLAB-DLL Element) or any commercial software featuring C-code generation (MATLAB-DLL Element). All the important functions of an electronic engine control device can be simulated.

23.1. MATLAB® API (Simulink)

Element Name:	**MATLAB® API: Simulink**	
Element Icon:		
Definition:	This element defines the properties of an API Interface between BOOST Hydsim and MATLAB® Simulink.	
Connecting pins:	standard pins: 0 special pins: 0 wire pins: 4	

Note: Currently the Variable-step solver does not support this element.

Note: To enable calling MATLAB® from **BOOST Hydsim** on Linux platforms, the location of the API shared libraries has to be defined and the path of **MATLAB®** executable (matlab.exe) has to be set. These settings have to be made in the same shell where **BOOST Hydsim** is started.

- In a C shell, the command for setting the library path is:

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

- In a C shell, the command for setting the MATLAB path is:

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

- In a Bourne shell, the commands for setting the library path are:

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

export LD_LIBRARY_PATH
```

- In a Bourne shell, the command for setting the MATLAB path is:

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

export PATH
```

where <MATLAB> is the **MATLAB®** root directory and $Arch is the system architecture (alpha, glnx86, sgi, sol2, hpux, hp700 or ibm_rs).

The environment variable LD_LIBRARY_PATH varies on several platforms:

HP700:	SHLIB_PATH
IBM RS/6000 :	LIBPATH

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

Note: **MATLAB® API: Simulink** element may have unlimited number of wire connections on each pin.

Note: In one **BOOST Hydsim** model up to 10 **MATLAB® API: Simulink** elements can be used. Each **Simulink** model must have a unique name.

23.1.1. Input Parameters

The Input Data Dialog consists of three main dialogs accessible from the Dialog element tree:

- **General** (shown in Figure 229)
- **Input Channels**
- **Output Channels**

General Input Dialog consists of **Control** and **User Vector** Input Data Dialogs.

23.1.1.1. MATLAB® Simulink: General Input Data

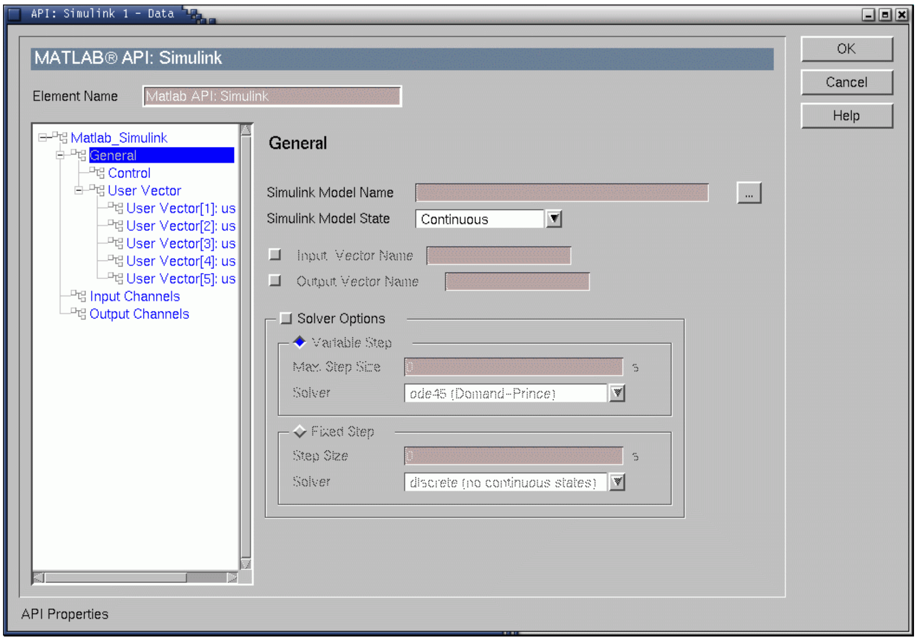


Figure 229: MATLAB® API: Simulink General Input Data Dialog

Description of input data:

Element name	The name of the element is specified as default.
Simulink Model Name	Specify the Simulink model name with the full path. Path may contain environmental variables, e.g. $(USER_HOME).
Simulink Model State	Specify the type of Simulink simulation: continuous or non-continuous. **Note:** If **Continuous** State is selected, BOOST Hydsim will use Final State of the previous Simulink calculation as Initial State for the next simulation.
Input Vector Name (optional)	Specify the name of input vector. It is necessary if **Constant** element is used for inputs in the Simulink model. **Note: Input vector** has to be disabled if **Input Port** elements are used as inputs in the Simulink model.
Output Vector Name (optional)	Specify the name of output vector. It is necessary if **To Workspace** element is used for outputs in the Simulink model. **Note: Output vector** has to be disabled if **Output Port** elements are used as outputs in the Simulink, model.

Solver Options (optional)	By activating this button, appropriate parameters for the Simulink Solver (type of incrementation, step size and solver type) can be set before starting the co-simulation.
Variable Step	Sets the variable step size for Simulink simulation.
Max.Step Size	Specify the largest time step to be used by the solver in Simulink simulation. If this parameter is set to 0, the default value is determined from the BOOST Hydsim calculation time step (t_{calc}) : $$Max.\ step\ size = \frac{t_{calc}}{50}$$
Solver	Specify type of ordinary differential equations (ODEs) to be set as Solver type for Simulink simulation with **Variable Step**. It is possible to select one of the following solvers: - discrete (no continuous states) - ode45 (Domand - Prince) - ode23 (Bogacki - Shampire) - ode113 (Adams) - ode15s (stiff/NDF) - ode 23s (stiff/Mod.Rosenbrock) - ode223t (mod. stif/Trapezoid) - ode 23tb (stiff/TR-BDF2)
Fixed Step	Sets the fixed step size for Simulink simulation.
Step Size	Specify the fixed time step to be used by the solver in Simulink simulation. If this parameter is set at 0, the default value is determined from the BOOST Hydsim calculation time step (t_{calc}) :: $$Step\ size = \frac{t_{calc}}{50}$$
Solver	Specify type of ordinary differential equations (ODEs) to be set as Solver type for Simulink simulation with **Fixed Step**. It is possible to select one of the following solvers: - discrete (no continuous states) - ode5 (Domand - Prince) - ode4 (Runge – Kutta) - ode3 (Bogacki – Shampine) - ode2 (Heun) - ode1 (Euler)

23.1.1.2. MATLAB® Simulink: Control Input Data

Description of input data:

Sampling Time	Specify sampling time at which MATLAB Input channel (Sensor) scans information from the connected BOOST Hydsim element (refer to Figure 233). **Note:** If **Sampling time** is not defined (zero), it is set to **BOOST Hydsim** time step. If **Sampling time** is smaller than **BOOST Hydsim** time step, it will be reset to **BOOST Hydsim** step and a respective **Warning** message will be issued.
Response Time	Specify time delay (from sampling event) after which MATLAB Output channel (Actuator) sends information to the connected BOOST Hydsim (refer to Figure 232. Response time is actually a reaction time of MATLAB real-time model (due to inertia etc.). **Note: Response time** must be smaller than or equal to the **Sampling time**. **Note:** If **Response time** is not defined (zero), it is set to **BOOST Hydsim** time step. If **Response time** is smaller than **BOOST Hydsim** time step, it will be reset to **BOOST Hydsim** step and a respective **Warning** message will be issued.
User-defined Range (optional)	This button activates **User-defined Range Table**
User-defined Range Table	This table serves to define independent BOOST Hydsim-MATLAB® co-simulation subintervals. Within these subintervals BOOST Hydsim-MATLAB® co-simulation will be activated. Specify start time in ascending order in the first column and stop time in ascending order in the second column. **Note:** Start and Stop time instants have to be arranged in the ascending order, i.e. each new start time must be greater than the previous stop time. **Note:** Maximum 15 subintervals can be used (15 rows in the User-defined Range table).

23.1.1.3. MATLAB® Simulink: Input Data of User Vector

Description of input data:

Vector Name	The name of the vector is specified as default.
Parameter (table)	Specify entries of a user vector. User vector entries can be viewed in Impress Chart window by pressing the **View** bar.

23.1.1.4. MATLAB® Simulink: Input Data of Input Channels

Description of input data:

#	Rows (input channels) are numbered automatically. Serial channel number implies an input port number in the Simulink model. **Note:** Maximum 10 Input Channels can be used within one MATLAB® API:Simulink element.
Type	Specify type of input parameter. There are four types available: • **External Value** • **External Table** • **Element** • **Global** According to the selected type of input, the corresponding table column(s) will be activated: 1. **External Value** enables **Value** column 2. **External Table** enables **External Table** input dialog 3. **Element** enables **Element, Type of Property** and **Property** columns (refer to Section 23.1.2 for the list of the available Input parameters) 4. **Global** enables **Property** column (refer *to* Section 23.1.3 for the list of Global parameters)
Value	Specify constant input parameters. Select **External Value** in **Type** pop-up menu to define constant input parameter and click in **Value** column. Type the value in SI units.
External Table	Select **External Table** in **Type** pop-up menu and click on **External Table[n]** in the Element dialog tree, where **n** is the row number of the selected channel. The **External Table** dialog will pop up. Data in 2nd column must be specified in **SI units**.
Element	Select an appropriate element which parameters will be used for input channels. Select **Element** in **Type** pop-up menu to define Element properties and click in the **Element** column to choose the appropriate element. List of available elements will pop up. It contains all BOOST Hydsim elements connected by wire connection to MATLAB® API:Simulink element.
Type of Property	Specify type of element property for the input parameters. Select appropriate type of property from the list in the pop-up menu. The list of type of properties depends on the selected element (refer to Section 23.1.2).
Property	Specify element property for the input parameters. Select appropriate property from the property list in the pop-up menu. The list of element properties is individual for each element (refer to Section 23.1.2).

23.1.1.5. MATLAB® Simulink: Input Data of Output Channels

Description of input data:

#	Rows (input channels) are numbered automatically. Serial channel number implies an output port number in the Simulink model. **Note:** Maximum 10 Output Channels can be used within one MATLAB® API:Simulink element.
Element	Select an appropriate element which parameters will be used for output channels. Click in the **Element** column to select the appropriate element. List of available elements will pop up. It contains those BOOST Hydsim elements which are connected by wire connection to MATLAB® API:Simulink element and which have at least one controllable parameter.
Type of Property	Specify type of element property for the output parameters. Select appropriate type of property from the list in the pop-up menu. The list of type of properties depends on the selected element (refer *to* Section 23.1.4*)*.
Property	Specify element property for the output parameters. Select appropriate property from the property list in the pop-up menu. The list of element properties is individual for each element (refer to Section 23.1.4).

23.1.2. Input element properties

There are three types of **input element properties**:

- **El. input parameters** (parameters from Input Data dialog)
- **El. output parameters** (parameters from the Output parameters dialog)
- **El. local fluid properties** (fluid properties in the element)

List of input and output is defined for each element. It can be accessed by the selection in **Element** column in Input Channels Table.

23.1.3. Global input parameters

Available **Global input parameters** are:

- time [s] (actual time)
- crank angle [rad] (actual crank angle)

23.1.4. Output element properties

There are three types **of output element properties**:

- **El. input parameters** (parameters from Input Data dialog)
- **El. output parameters** (parameters from the Output parameters dialog)
- **El. local fluid properties** (fluid properties in the element)

List of input and output is defined for each element. It can be accessed by the selection in **Element** column in the Output Channels Table.

23.1.5. Initial Conditions

Initial conditions cannot be specified for **MATLAB® API:Simulink** element.

23.1.6. Modifiable Parameters

Modifiable parameters cannot be specified for **MATLAB® API:Simulink** element.

23.1.7. Output Parameters

Path: **Element | Store Results**

The **Output Parameters** Dialog of **MATLAB® API: Simulink** is shown in Figure 230.

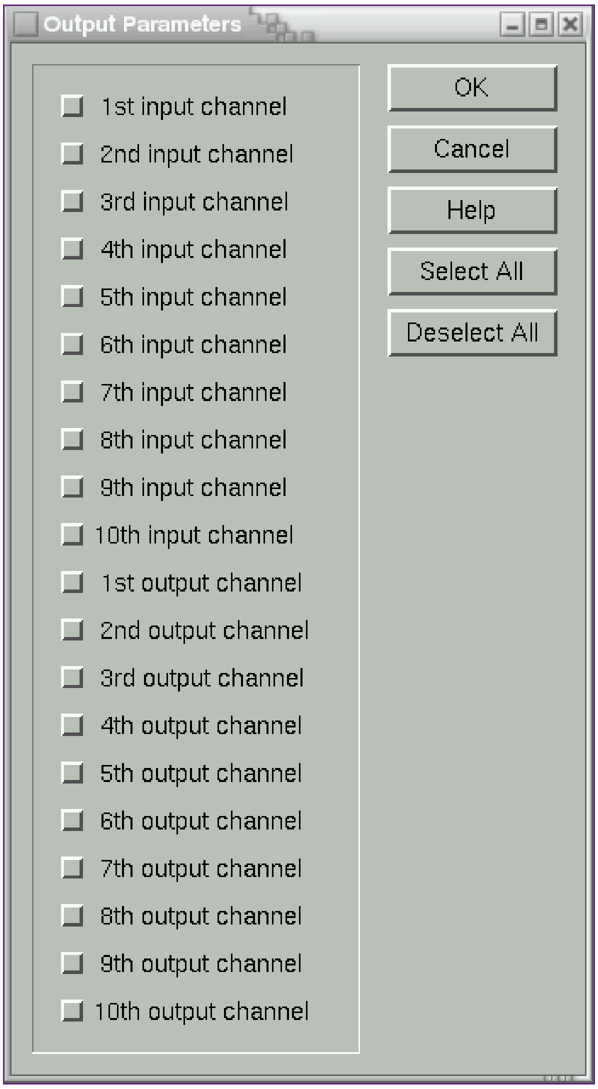


Figure 230: Output Parameters of MATLAB® API:Simulink Dialog

To activate output parameter, the option box in front of output parameter has to be checked.

Description of output parameters:

x-th input channel	Nonzero only if selected input channel is specified in **Input Channels** Dialog of **MATLAB® API: Simulink** element.
x-th output channel	Nonzero only if selected output channel is specified in **Output Channels** Dialog of **MATLAB® API: Simulink** element.

23.1.8. Co-simulation

BOOST Hydsim – MATLAB® API co-simulation is activated from the 2nd calculation time step. If the **User-defined Range** table is disabled, co-simulation will be performed from the beginning till end of the **BOOST Hydsim** calculation. If the **User-defined Range** table is defined, then co-simulation will be performed only within the specified range (subintervals).

If **Sampling time** and **Response time** are not specified, co-simulation will be performed with default values, i.e. at the each **BOOST Hydsim** calculation step as shown in Figure 231. Otherwise (**Sampling time** and/or **Response time** are defined by the user) co-simulation will be performed according to the rules explained in Figure 232, Figure 233 and Figure 234.

☞ **Note:** If **Sampling** and/or **Response Time** is not a multiple of **BOOST Hydsim** time step, they will be automatically adjusted and a corresponding **Warning** message will be issued.

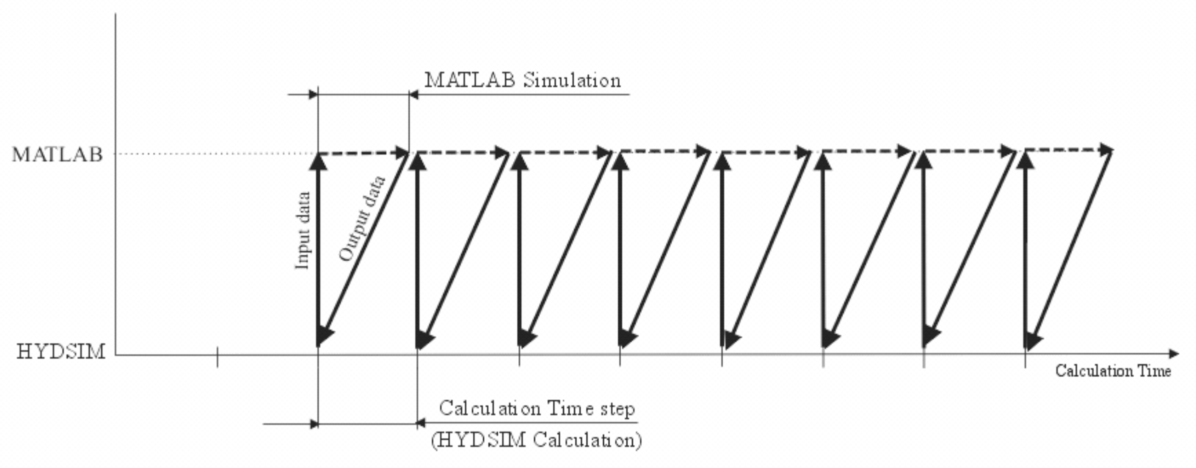


Figure 231: Co-simulation with Default Sampling and Response Time

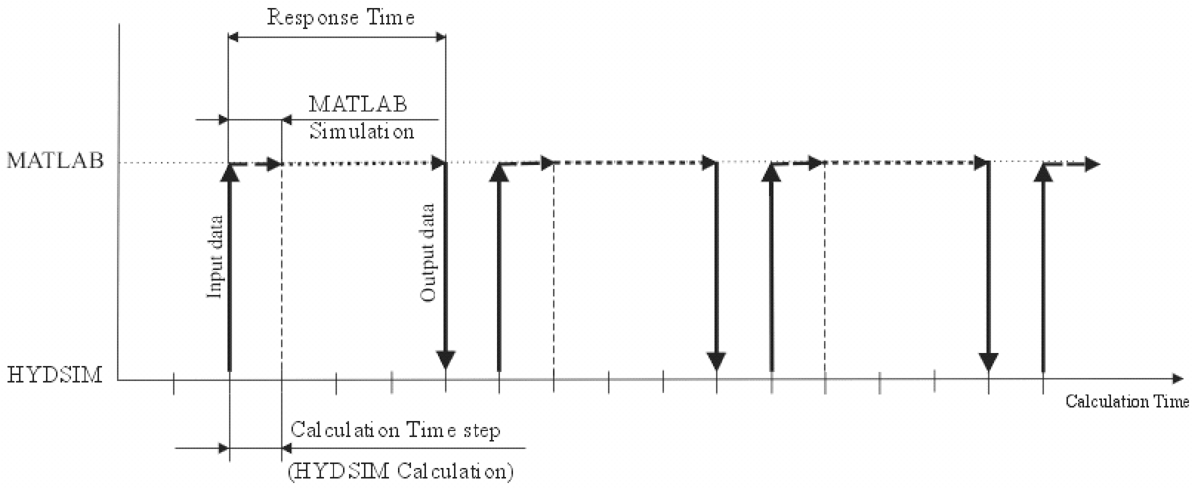


Figure 232: Co-simulation with Default Sampling and User-defined Response Time

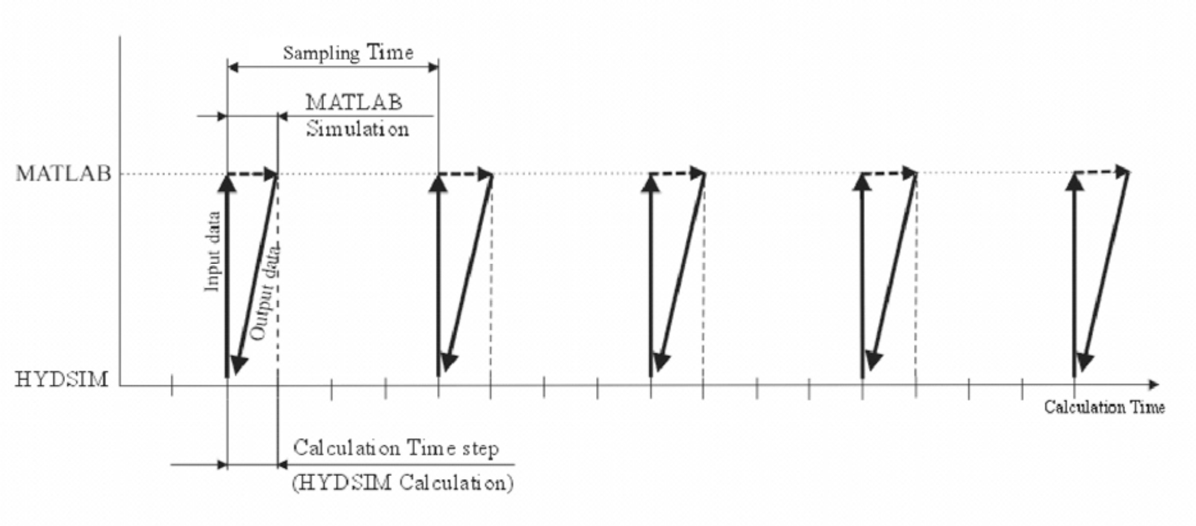


Figure 233: Co-simulation with User-defined Sampling and Default Response Time

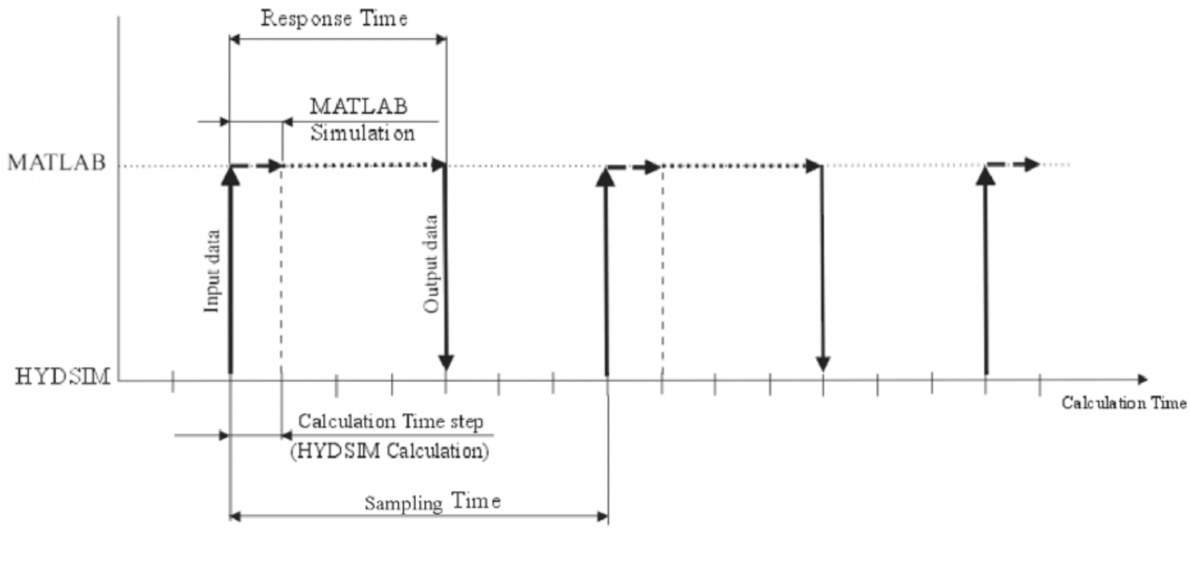
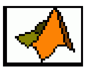
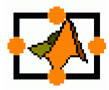


Figure 234: Co-simulation with User-defined Sampling and Response Time

23.2. MATLAB® API: m-function

Element Name:	**MATLAB® API: m-function**	
Element Icon:		
Definition:	This element defines the properties of an API Interface between BOOST Hydsim an MATLAB® m-function.	
Connecting pins:	standard pins: 0 special pins: 0 wire pins: 4	

Note: Currently the Variable-step solver does not support this element.

Note: **MATLAB® API: m-function** element may have unlimited number of wire connections on each pin.

Note: In one **BOOST Hydsim** model up to 10 **MATLAB® API: m-function** elements can be used. Each **m-function** model must have a unique name.

23.2.1. Input Parameters

The Input Data Dialog consists of three main dialogs accessible from the Dialog element tree:

- **General**
- **Input Channels**

- **Output Channels**

General Input Dialog consists of **Control** (shown in Figure 235) and **User Vector** Input Data Dialogs.

23.2.1.1. MATLAB® m-function: General Input Data

Description of input data:

Element name	The name of the element is specified as default.
m-function Name	Specify the m-function file name with the full path. Path may contain environmental variables, e.g. $(USER_HOME).
Input Vector Name	Specify name of the input vector used in the m-function.
Output Vector Name	Specify name of the output vector used in the m-function.

23.2.1.2. MATLAB® m-function: Control Input Data

The same as for MATLAB® API: Simulink (refer to Section 23.1.1.2).

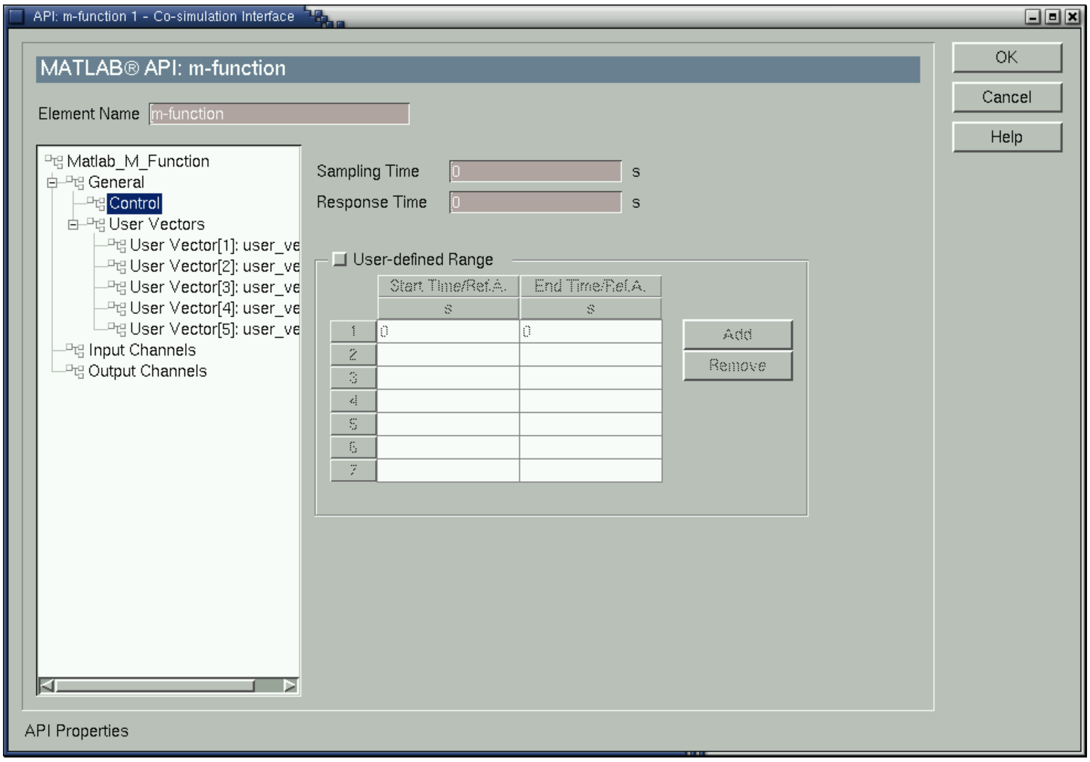


Figure 235: Control Input Data Dialog of MATLAB® API: m-function

23.2.1.3. MATLAB® m-function: Input Data of User Vector

Input data dialog of a user vector is identical to the respective dialog of MATLAB® API: Simulink element (refer to Section 23.1.1.3).

23.2.1.4. MATLAB® m-function: Input Data of Input Channels

Input data dialog of input channels is identical to the respective dialog of MATLAB® API: Simulink element (refer to Section 23.1.1.4).

23.2.1.5. MATLAB® m-function: Input Data of Output Channels

Input data dialog of output channels is identical to the respective dialog of MATLAB® API: Simulink element (refer to Section 23.1.1.5).

23.2.2. Input Element Properties

Input element properties are identical to those of MATLAB® API: Simulink (refer to Section **23.1.2**).

23.2.3. Global Input Parameters

Global input properties are identical to those of MATLAB® API: Simulink (refer to Section **23.1.3**).

23.2.4. Output Element Properties

Output element properties are identical to those of MATLAB® API: Simulink (refer to Section **23.1.4**).

23.2.5. Initial Conditions

No initial conditions can be specified for **MATLAB® API: m-function** element.

23.2.6. Modifiable Parameters

Modifiable parameters cannot be specified for **MATLAB® API: m-function** element.

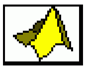
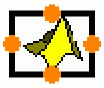
23.2.7. Output Parameters

The dialog of output parameters is identical to the respective dialog of MATLAB® API: Simulink element (refer to Section 23.1.7).

23.2.8. Co-simulation

Co-simulation procedure between BOOST Hydsim and MATLAB® API: m-function is the same as the co-simulation between BOOST Hydsim and MATLAB® API: Simulink (refer to Section 23.1.8).

23.3. MATLAB® DLL

Element Name:	**MATLAB® DLL**	
Element Icon:		
Definition:	This element defines the properties of an Interface between BOOST Hydsim and MATLAB® DLL.	
Connecting pins:	standard pins: 0 special pins: 0 wire pins: 4	

Note: Currently the Variable-step solver does not support this element.

Note: **MATLAB® DLL** is a Dynamically Linked Library generated from the **Simulink** model by **MATLAB® Real-Time Workshop**.

Note: **MATLAB® DLL** element may have unlimited number of wire connections on each pin.

Note: In one **BOOST Hydsim** model up to 10 **MATLAB® DLL** elements can be used. Each **MATLAB® DLL** file must have a unique name.

23.3.1. Real Time Workshop

The following procedure may be applied for the generation of a DLL using the MATLAB® Real Time Workshop tool. This procedure works with MATLAB® v5.3, v6.0, v6.1, v6.5 and higher.

Executing the command `mex -setup` in the MATLAB® command line opens a menu where the C/C++ compiler for the DLL generation has to be selected.

Depending on the MATLAB® version, the following path has to be added to the MATLAB® /Simulink path:

MATLAB® V5.3: `BOOST Hydsim_HOME..\..\matlab\v5.3`

By executing the MATLAB® command '`addpath`' ('path to add')

MATLAB® V6.0 and higher: `BOOST Hydsim_HOME..\..\matlab\v6.x`

via the MATLAB® GUI (**File | Set Path** select **Add Folder** and **SAVE**)

Furthermore, the environmental variable BOOST Hydsim_HOME has to be defined. It has to point to the directory where the `BOOST` Hydsim executable is stored, i.e.:

```
$(AVLAST_HOME)\BOOST Hydsim\v4.5\bin\$(PLATFORM)
```

1. Solver

Open **Solver** window from **Simulation|Simulation Parameters** pull-down menu and set the parameters as shown in Figure 236.

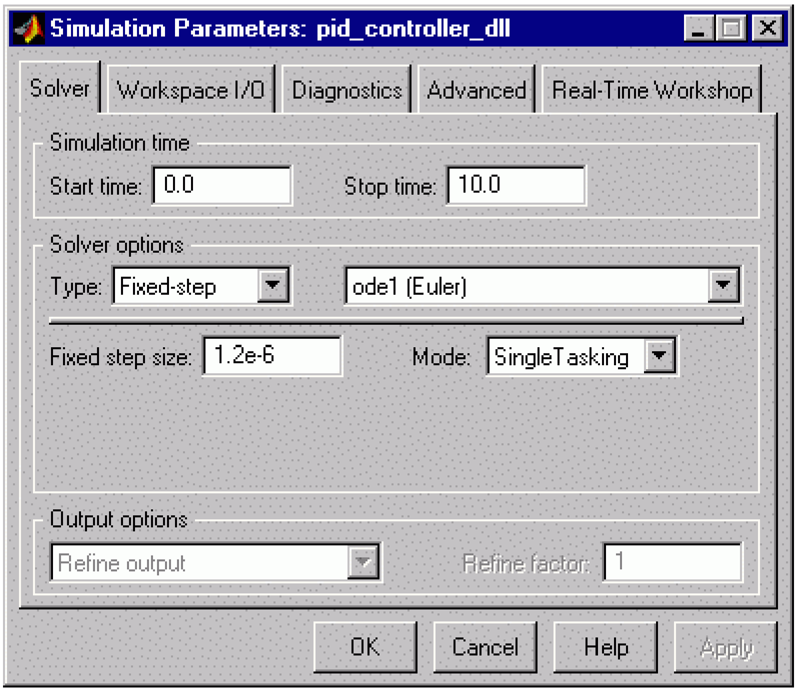


Figure 236: SIMULINK Solver Settings

In the Solver window (accessible from Simulation Parameters Dialog) the `incrementation` and `Solver type` have to be defined. The incrementation has to be constant (Fixed-step) and adapted to the selected calculation time step of BOOST Hydsim.

Note: **Incrementation** must be greater than or equal to the BOOST Hydsim calculation step. The values of **Start** and **Stop Time** have no meaning because these are set by BOOST Hydsim.

Note: If no integrators, memory blocks etc. are used in the Simulink model, the specified value of the **Fixed step size** is ignored: it is automatically reset to the BOOST Hydsim time step. If DLL incrementation is not a multiple of the BOOST Hydsim calculation time step, it will be adjusted to the closest (higher) multiple of the BOOST Hydsim step.

2. Workspace I/O

In the **Workspace I/O** window (refer to Figure 237) **Input** and **Initial state** parameters have to be disabled.

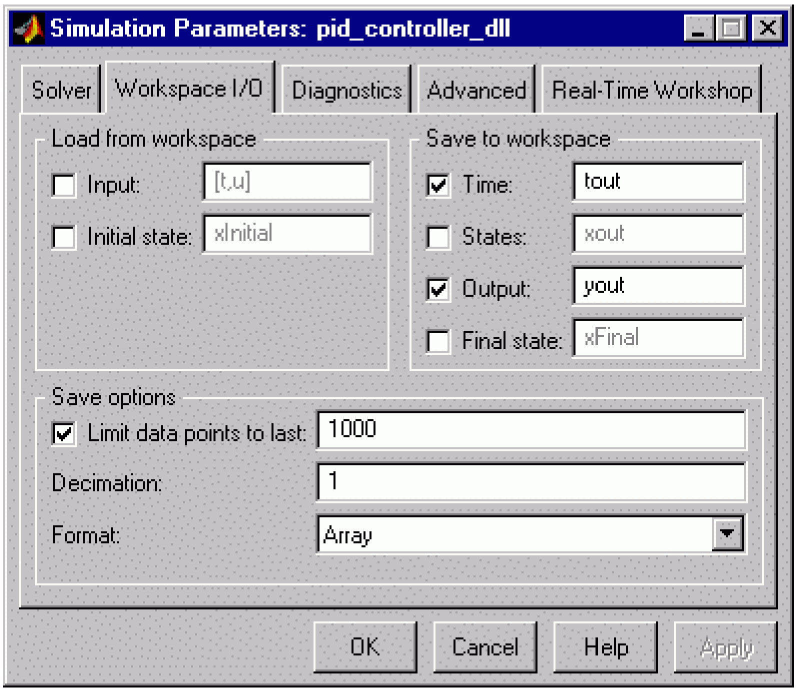


Figure 237: SIMULINK Workspace Settings

In the Workspace I/O window certain data of the SIMULINK™ block diagram can be written out to a MATLAB® readable file which is characterized by the extension MAT. After running the simulation this MAT file can be loaded into the Workspace from MATLAB®. This file has the same name as the Simulink model.

The actual time **Time**, the state variables **States** or the output values **Output** etc. can be selected for storing. The **Output** vector contains all output values. If `Scope` block or `To Workspace` block is used, then this data will also be written onto the MAT file. If none of the possible items are selected and no blocks for storing are used in the block diagram, MAT file will not be produced. **Format** option determines whether the values will be recorded in a simple matrix or structured format (with or without time).

3. Real time Workshop

In the **Real-Time Workshop** window (refer to Figure 238) the settings for the DLL generation have to be defined.

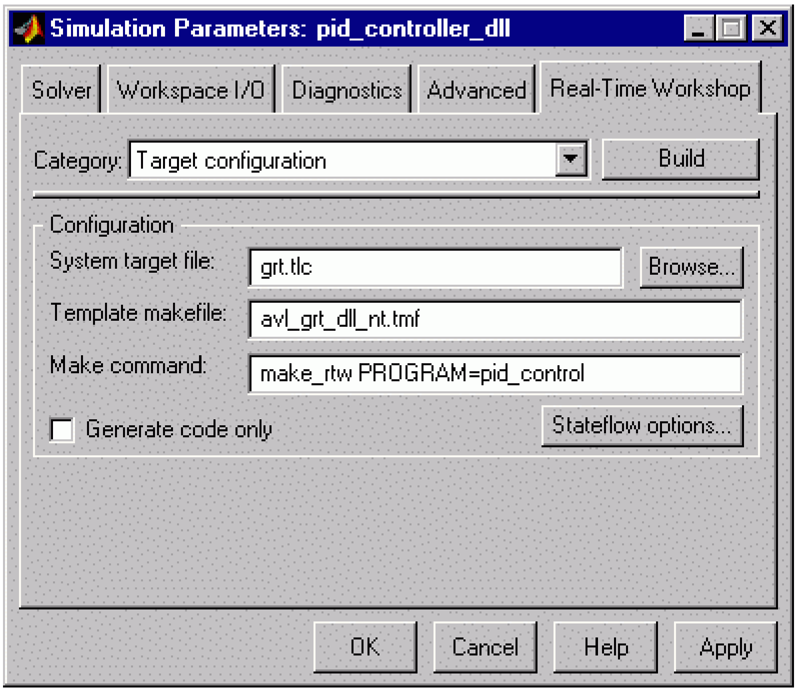


Figure 238: SIMULINK Real-Time Workshop Settings

System Target file defines the type of the code which can be produced. Enter `grt.tlc` (general real-time).

Template makefile defines the name of the template file from which the new **makefile** is generated. For generating of the desired DLL file using the VC++ V6.0 compiler, template file `avl_grt_dll_nt.tmf` (on Windows) or `avl_grt_dll_unix.tmf` (on Linux) has to be loaded. If its path is not added to the MATLAB®/SIMULINK™ path (using **Set Path** command), this file has to be placed either into the directory `%MATLAB ROOT%\rtw\c\grt` or into the current working directory. Otherwise the complete path name has to be entered.

For **Make command** the text `make_rtw PROGRAM=pid_control` has to be entered. `PROGRAM=pid_control` defines the DLL file name. Alternatively the model name is used and an underscore is placed in front. If the DLL file name has to be same as the Simulink model (MDL file) name, an underscore has to be placed in front (refer to the following table). Otherwise no valid C-MEX DLL will be generated, as the MDL file in MATLAB® could not be opened any longer, because it is always checked first whether a DLL file with the same name is available.

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`

Select **Build** to generate a C-code. The `Makefile` will be produced, `nmake.exe` will be executed with this `Makefile` and the object files will be linked to the DLL.

23.3.2. Input Parameters

The input data dialog consists of three main dialogs accessible from the dialog element tree:

- **General**
- **Input Channels**
- **Output Channels**

23.3.2.1. MATLAB® DLL: General Input Data

Description of input data for **MATLAB® DLL**:

Element name	The name of the element is specified as default.
DLL Model Name	Specify the DLL file name with the full path. Path may contain environmental variables, e.g. $(USER_HOME).

23.3.2.2. MATLAB® DLL: Input Data of Input Channels

Input data dialog of input channels is identical to the respective dialog of MATLAB® API: Simulink element (refer to Section 23.1.1.4).

23.3.2.3. MATLAB® DLL: Input Data of Output Channels

Input data dialog of output channels is identical to the respective dialog of MATLAB® API: Simulink element (refer to Section 23.1.1.5).

23.3.3. Input Element Properties

Input element properties are identical to those of MATLAB® API: Simulink (refer to Section 23.1.2).

23.3.4. Global Input Parameters

Global input properties are identical to those of MATLAB® API: Simulink (refer to Section 23.1.3).

23.3.5. Output Element Properties

Output element properties are identical to those of MATLAB® API: Simulink (refer to Section 23.1.4).

23.3.6. Initial Conditions

Initial conditions cannot be specified for **MATLAB® DLL** element.

23.3.7. Modifiable Parameters

Modifiable parameters cannot be specified for **MATLAB® DLL** element.

23.3.8. Output Parameters

Path: **Element** | **Store Results**

The **Output Parameters** Dialog of **MATLAB® DLL** is shown in Figure 239.

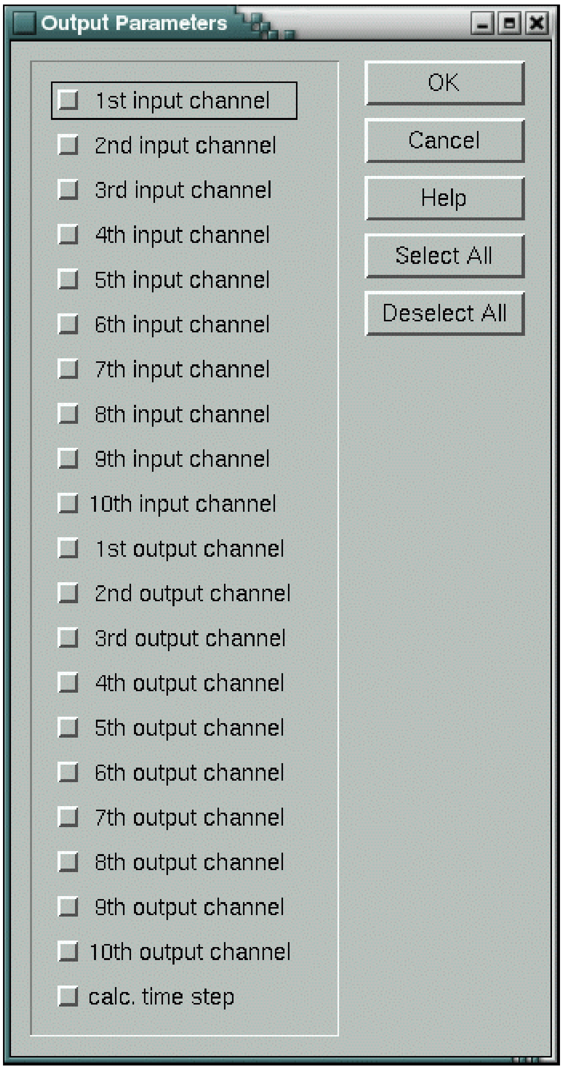


Figure 239: Output Parameters of MATLAB® DLL Dialog

To activate output parameter, the option box in front of output parameter has to be checked

Description of output parameters:

x-th input channel	Nonzero only if selected input channel is specified in **Input Channels** Dialog of **MATLAB® DLL** element (refer to Section 23.3.2.2).
x-th output channel	Nonzero only if selected output channel is specified in **Output Channels** Dialog of **MATLAB® DLL** element (refer to Section 23.3.2.3)
model calc. time step	Incrementation of MATLAB® DLL

23.3.9. Simulation

BOOST Hydsim – MATLAB® DLL Interface is activated from the 2nd calculation time step. Simulation with DLL is performed from the beginning till end of the **BOOST Hydsim** calculation. Data exchange procedure depends on the `Incrementation` specified while generating the DLL from the Simulink model as shown in Figure 240 and Figure 241. For more detailed information on creating the MATLAB® DLL refer to the **BOOST Hydsim Primer**.

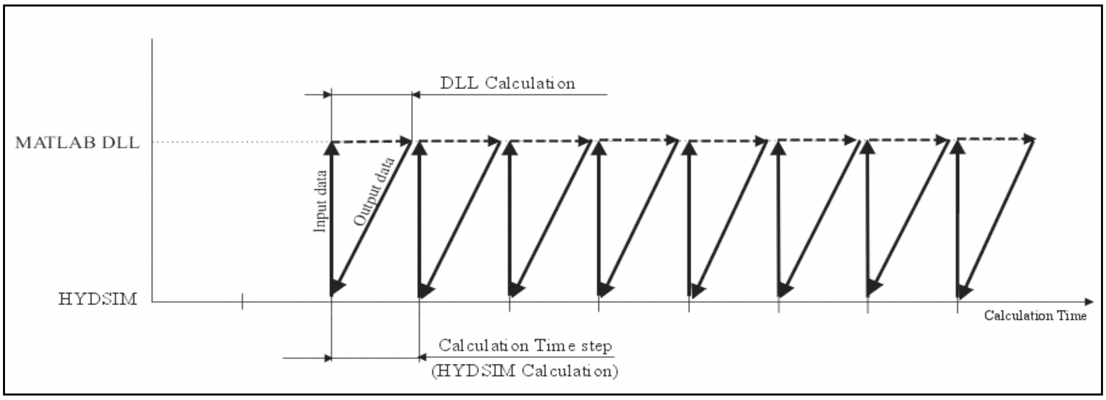


Figure 240: Simulation with DLL Incrementation Equal to BOOST Hydsim Time Step

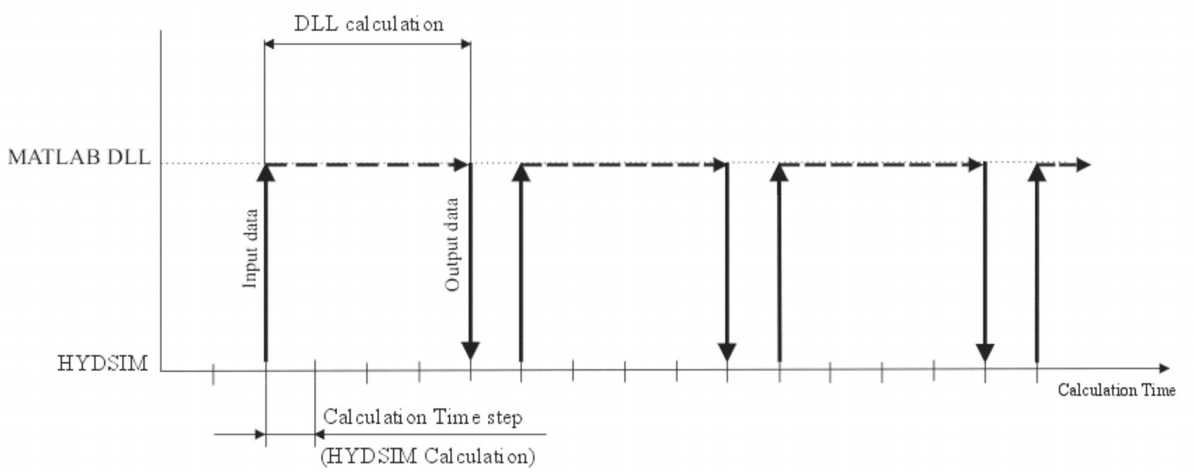
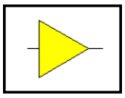
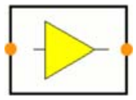


Figure 241: Simulation with DLL Incrementation Greater than BOOST Hydsim Time Step

23.4. PID Controller

Element Name:	**PID Controller**	
Element Icon:		
Definition:	This element serves to define a feedback controller that uses a selected output parameter from a selected element (sensor), compares its value with a guiding (target) value, which can be either external or from another element, and adjusts a selected input parameter of a controlled element (actuator) using a combination of proportional, integral and derivative control.	
Connecting Pins:	standard pins: 0 special pins: 0 wire pins: 2	

Block diagram of a PID Controller element is shown in Figure 242. PID controller calculates an "error" value as the difference between a measured process variable and a desired target (set-point). The controller attempts to minimize the error by adjusting the process control input. However, for best performance, the PID parameters used in the calculation have to be tuned according to the physics of the system. The PID controller calculation involves three separate parameters: the proportional, integral and derivative gains, denoted P, I, and D. The proportional gain determines the reaction to the current error, the integral gain determines the reaction based on the sum of recent errors, and the derivative gain determines the reaction based on the rate at which the error has been changing. The weighted sum of these three actions is used to adjust the process via control element such as the position of a control valve.

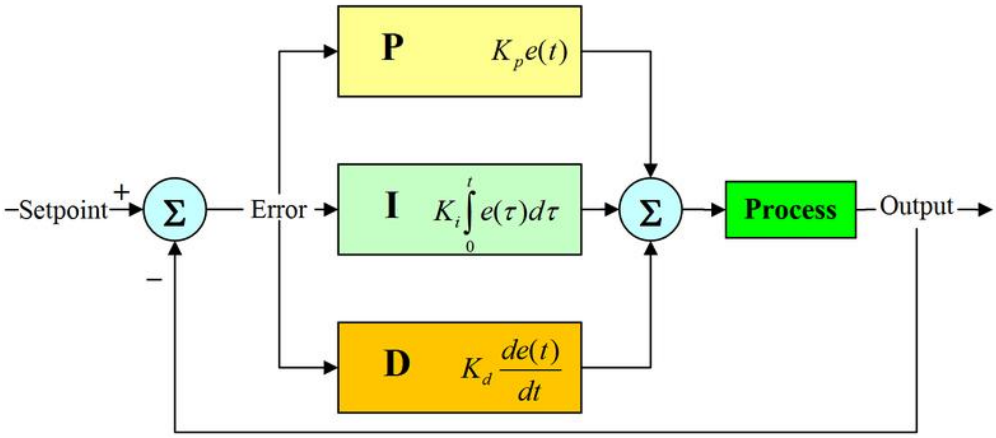


Figure 242: Block Diagram of PID Controller

23.4.1. Input Parameters

The Input Data Dialog consists of three main dialogs accessible from the Dialog element tree:

- **General**
- **Channels**
- **External Table (optional)**

23.4.1.1. PID Controller : General Data

Description of input data:

Element Name	The name of the element is specified as default.
Gain	In this block the gains or coefficients for each of the three controller terms are specified.
Proportional Gain	Specify the gain for proportional term of controller.
Integral Gain	Specify the gain for integral term of controller.
Derivative Gain	Specify the gain for derivative term of controller.
Offset	Specify the offset between sensor value and guiding value. **Note**: Offset unit is automatically adjusted to guiding value unit.
Interaction Step	Select the interaction step type from the pull-down menu.
Every Timestep	Selecting this option implies that the controller receives sensor values on every time step and the actuator responds immediately.
Specified Timestep	Selecting this option will enable the user to specify communication interval for PID larger than time step, and to input the delay of actuator response.
Sampling Time	When Specified Timestep option is selected, specify the time interval for communication of PID Controller with sensor element. This interval has to be larger than the time step, otherwise the time step will be used as the default Sampling Time.
Response time	When Specified Timestep option is selected, specify the time delay between sensor signal reception and the actuator response. This interval has to be larger than time step, otherwise the time step will be used as the default Response Time.

23.4.1.2. PID Controller : Channels

Description of input data:

Sensor Element	Select an appropriate element which parameters will be used for input channels. **Note**: Only elements which are connected to the PID Controller element via wire connection will be visible on the list.
Sensor Channel	Specify element property for the input parameters. Select the appropriate property from the property list in the pop-up menu. The list of available element properties is individual for each element.

Integral Minimum Value (optional)	Specify the minimum value of integral term, below which it cannot drop.
Integral Maximum Value (optional)	Specify the maximum value of integral term, above which it cannot rise.
Guiding Value Type	Specify type of input parameter. There are three types available: • **External Value** • **External Table** • **Element** According to the selected type of input, the corresponding table column(s) will be activated: 1. **External Value** enables **Value** column 2. **External Table** enables **External Table** input dialog 3. **Element** enables **Element**, **Guiding Value Channel** columns
Value	Specify constant input parameters. Select **External Value** in **Type** pop-up menu to define constant input parameter and click in **Value** column. **Note**: Selected value must be specified in SI units.
External Table	Select **External Table** in **Type** pop-up menu and click on **External Table** in the Element dialog tree. The **External Table** dialog will pop up. **Note**: Data in 2nd column must be specified in SI units.
Element	Select an appropriate element which parameters will be used for input channels. Select **Element** in **Type** pop-up menu to define Element properties and click in the **Element** column to choose the appropriate element. List of available elements will pop up. . **Note**: Only elements which are connected to the PID Controller element via a wire connection will be visible on the list.
Guiding Value Channel	Specify element property for the input parameters. Select appropriate property from the property list in the pop-up menu. The list of element properties is individual for each element.
Actuator element	Select an appropriate element which parameters will be adjusted by PID controller. **Note**: Only elements which are connected to the PID Controller element via a wire connection will be visible on the list.

Actuator channel	Specify element property for the output parameters. Select appropriate property from the property list in the pop-up menu. The list of element properties is individual for each element.
Initial Value	Specify the initial value for output (control variable).
Minimum Value	Specify the minimum value for output (control variable), below which it cannot drop.
Maximum Value	Specify the maximum value for output (control variable), above which it cannot rise.
	Note: Units are automatically adjusted to actuator channel unit.

23.4.2. Output Parameters

Path: **Element | Store Results**

To activate the output parameter, the check box on the left of the parameter name has to be checked.

Sensor Value	Actual value of Sensor parameter.
Guding Value	Actual value of Guiding parameter.
Actuator Value	Actual value of Actuator parameter.

23.4.3. Theoretical Background

As shown in Figure 242, P, I and D stand for Proportional,-Integral-and Derivative. This is a type of feedback controller whose output, an actuator control variable (AV), is generally based on the error (e) between a guiding value (GV) and certain sensor value (SV). Each element of the PID controller refers to a particular action imposed on the error. If a positive change in the PID actuator output causes a positive change to the sensor value, the controller gains should be positive. If a positive change to the actuator output causes a negative change to the sensor value, then the controller gains should be negative.

The PID control scheme is named after its three correcting terms, namely:

- **Proportional**:
  Error multiplied by a gain, *Kp*. This is an adjustable amplifier. In many systems Kp is responsible for process stability: if *Kp* is too low, the SV can drift away; if *Kp* is too high, the SV can oscillate. The equation of proportional term is given by:

$$P_i = K_p e_i(t) \tag{480}$$

- **Integral**:
  The integral of error multiplied by a gain, *Ki*. In many systems *Ki* is responsible for driving error to zero. However, setting *Ki* too high often leads to oscillations or instability, integrator windup or actuator saturation. The equation of integral term is as follows:

$$I_i = K_i \int_0^t e_k(\tau) \cdot d\tau \tag{481}$$

- **Derivative:**
  The rate of change of error multiplied by a gain, *Kd*. In many systems *Kd* is responsible for system response: if *Kd* is too high, the SV will oscillate; if *Kd* is too low, the SV will respond sluggishly. It should also be noted that derivative action amplifies any noise in the error signal. The equation of derivative term is given by:

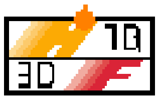
$$D_i = K_d \frac{d}{dt} e_i(t) \tag{482}$$

Tuning of a PID controller involves the adjustment of gains *Kp*, *Ki*, and *Kd* to achieve some user-defined "optimal" character of system response. The output of PID controller is calculated as the sum of proportional, integral and derivative terms:

$$AV_i(t) = P_i + I_i + D_i = K_p e_i(t) + K_i \int_0^t e_k(\tau) \cdot d\tau + K_d \frac{d}{dt} e_i(t) \tag{483}$$

24. EXTERNAL

24.1. Fire Link

Element Name:	**Fire Link (Nozzle)**	
Element Icon:		
Definition:	This element defines the properties of an Interface between BOOST Hydsim and FIRE (nozzle inner flow simulation).	
Connecting pins:	standard pins: 0 special pins: 0 wire pins: 1	

Note: Currently the Variable-step solver does not support this element.

Note: **Fire Link** element may only have one wire connection on each pin.

Note: In one **BOOST Hydsim** model only one **Fire Link** element can be used.

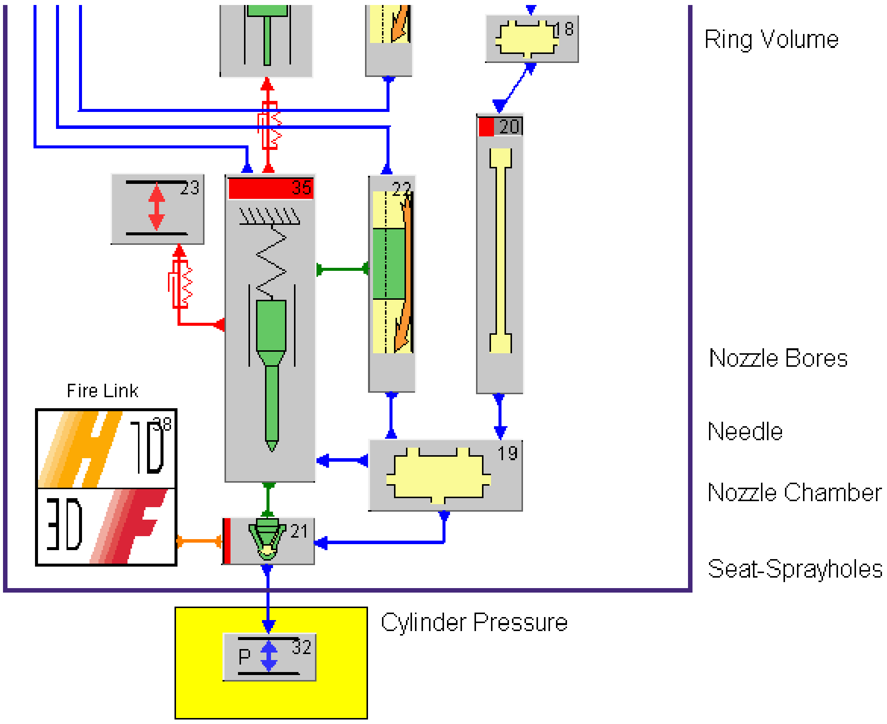


Figure 243: Hydsim model with Fire Link element

24.1.1. Input Parameters

Description of input data for the **Fire Link** element.

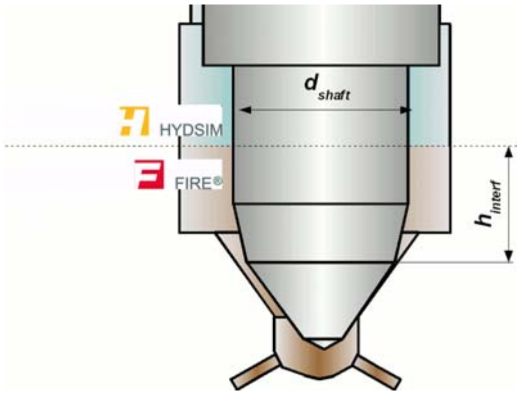
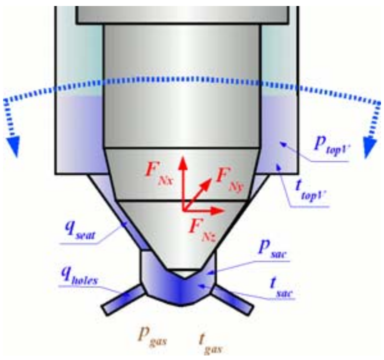
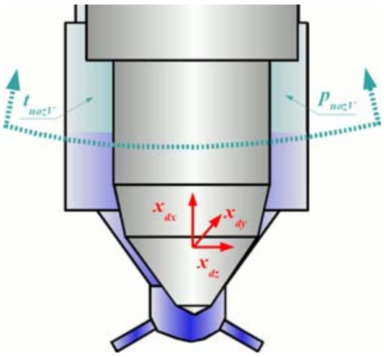
<table>
<tr><td>Element name</td><td colspan="2">The name of the element is specified as default.</td></tr>
<tr><td colspan="3">GENERAL</td></tr>
<tr><td>Interface height from needle seat</td><td colspan="2">Specify interface height from needle seat h_{interf} .
(refer to Figure 188)</td></tr>
<tr><td>Needle shaft diameter at inteface</td><td colspan="2">Specify needle shaft diameter at inteface d_{shaftf} .
(refer to Figure 188)</td></tr>
<tr><td colspan="3">

Figure 244: Interface between FIRE and Hydsim</td></tr>
<tr><td colspan="3">Data output to files</td></tr>
<tr><td rowspan="2">Write Fire data to file</td><td colspan="2">Specify file name (with the full path) for storing FIRE data transferred to BOOST Hydsim via ACCI.
Path may contain environmental variables, e.g. $(USER_HOME).</td></tr>
<tr><td colspan="2">List of variables which FIRE transfers to BOOST Hydsim:
needle_force(1) ········ Component of F_{hyd} actíng onto needle tip in x-direction
needle_force(2) ········ Component of F_{hyd} acting onto needle tip in y-direction
needle_force(3) ········ Component of F_{hyd} acting onto needle tip in z-direction
q_seat ························ Mass flow rate through needle seat
q_holes ······················ Mass flow rate through spray holes
p_topV ······················ Average pressure below interface line (FIRE side)
T_topV ······················ Average temperature below interface line (FIRE side)
p_sac ························ Pressure in sac volume (user-selected mesh area)
T_sac ························ Temperature in sac volume (user-selected mesh area)
p_gas ························ Pressure in combustion chamber (boundary)
T_gas ························ Temperature in combustion chamber (boundary)

</td></tr>
</table>

Write Hydsim data to file	Specify file name (with the full path) for storing BOOST Hydsim data transferred to FIRE via ACCI. Path may contain environmental variables, e.g. $(USER_HOME).
	List of variables which BOOST Hydsim transfers to FIRE: displacement(1) .... Needle translation in x-direction displacement(2) .... Needle translation in y-direction displacement(3) .... Needle translation in z-direction p_nozV .... Pressure in nozzle chamber T_nozV .... Temperature in nozzle chamber 

Note: Output files are optional (for data control only). Data exchange is based on TCP/IP protocol.

CONTROL

Exchange time/angle step (open needle)	Specify time/angle step for data exchange with FIRE at open needle (refer to Figure 245).

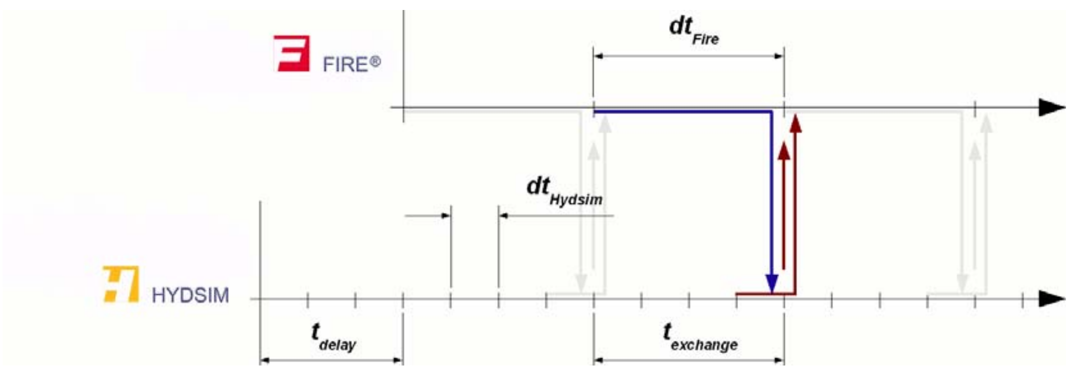


Figure 245: Exchange time/angle step

START/STOP INSTANT(S)

Needle lift-dependent

Needle lift tolerance (open/close)	Specify needle lift value at which BOOST Hydsim will start co-simulation with FIRE (x_{Ntol}).

Time delay from calculation start	Specify time interval (t_{delay}) from start for standalone BOOST Hydsim calculation. During it BOOST Hydsim will not activate co-simulation with FIRE no matter if needle lift reaches the tolerance or not.
Exchange step at closed needle	Specify time step for data exchange with FIRE at closed needle. If not defined (inactive), the exchange step at open needle will be used

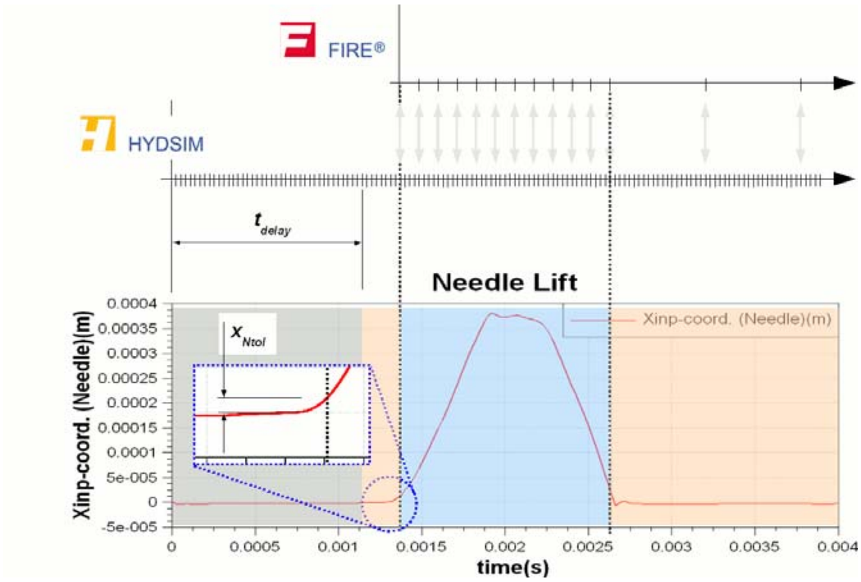


Figure 246: Needle Lift Tolerance (Hydsim -FIRE data exchange)

☞	**Note:** Data exchange starts when needle lift reaches tolerance first time (from below). If needle lift drops below the tolerance, exchange step at closed needle is used until the lift reaches its tolerance again. **Note:** With this option, data exchange with FIRE (once started) is active until BOOST Hydsim calculation end. **Note:** For continuing calculation from the end or previously saved time instant, **Restart** option can be employed. Time instants for saving Re-start file are controlled by FIRE (BOOST Hydsim option “Save interval for restart” is diasbled in this case). **Note:** Exchange time/angle step and Time delay from calculation start are “Simulation Period (Cranktrain)” variables related to Engine (crankshaft) speed.

<u>User-defined (closed needle)</u>	
User-defined Range	Specify time/crank angle data for the data exchange start/end.

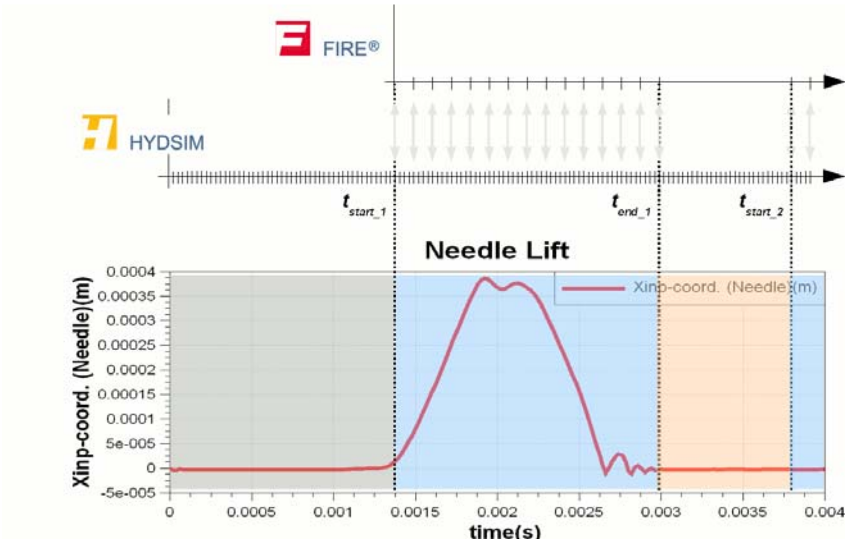


Figure 247: User-defined Range (Hydsim -FIRE data exchange)

☞	**Note:** Data exchange starts at 1st Start time of the first interval and ends at End Time of the last user-defined interval. Between two neighboring intervals a single (usually large) exchange step is used. **Note:** For continuing coupled calculation from the last End Time or previously saved time instant, **Restart** option can be employed. Time instants for saving Re-start file are controlled by FIRE.

24.1.2. Initial Conditions

Path: **Element | Initials**

Initial conditions cannot be specified for **Fire Link** element.

24.1.3. Modifiable Parameters

Path: **Element | Modify**

Modifiable parameters cannot be specified for **Fire Link** element.

24.1.4. Output Parameters

Path: **Element | Store Results**

Output Parameters cannot be specified for **Fire Link** element.

24.1.5. Communication between BOOST Hydsim and FIRE

The layer-structure of the coupling module

The coupling is performed by a separate software module, called ACCI (AVL Code Coupling Interface) integrated in FIRE and BOOST Hydsim. This module provides the following functionality:

- data-transfer and buffering between the different processes
- time control and synchronization of the processes

The data transfer is based on the TCP/IP protocol (socket mechanism). The coupled simulation processes may run on the same or on different computers, as long as they are in the same network.

The code coupling module consists of three different software-layers: the Communication Layer (ACI) at the lowest level, the Application Layer (ACCI), which provides all the basic functionality for co-simulations, and the BOOST Hydsim -ACCI Interface.

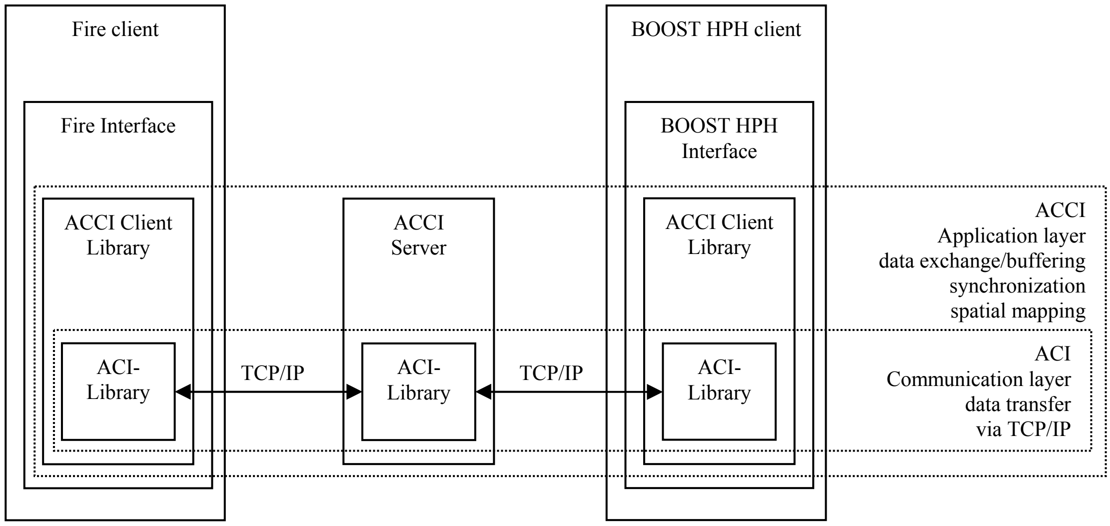


Figure 248: Software layers of the code coupling module in BOOST Hydsim

For ACCI, the host-name and port number (e.g. 7777) have to be defined at start. From FIRE side, it is usually specified in the CFD-WM calculation wizard. Similar wizard is incorporated in AWS/BOOST Hydsim GUI within Simulation/ACCI Interface/FIRE Nozzle dialog. In this way, the user has to type in the same host-name and port number in both client wizards (twice).

FIRE gets from BOOST Hydsim the needle motion vector $\boldsymbol{d}$ and boundary pressure and temperature $\boldsymbol{p_{nozV}}$ and $\boldsymbol{T_{nozV}}$ for its actual time step $\boldsymbol{\Delta t}$. In the opposite direction, BOOST Hydsim receives from FIRE the needle force vector $\boldsymbol{F}$, flow rates $\boldsymbol{Q_{seat}}$ and $\boldsymbol{Q_{holes}}$, sac pressure and temperature $\boldsymbol{p_{sac}}$ and $\boldsymbol{T_{sac}}$ and cylinder gas pressure and temperature $\boldsymbol{p_{gas}}$ and $\boldsymbol{T_{gas}}$. The needle force is computed by FIRE using a 2D-Result-Formula which integrates the pressure over the needle surface, according to the specified integration height $\boldsymbol{h_{inter}}$ above the needle coordinate system center. Value of height $\boldsymbol{h_{inter}}$ has to be correctly specified in Fire Link element dialog in BOOST Hydsim.

Communication layer - ACI

The communication-layer (ACI, abbreviation for AVL-AST Communication Interface) provides the functionality necessary for transferring data between the co-simulating processes. A client-server-model is used, i.e. the processes do not communicate directly with each other, but through a server program. The server runs as a separate process that waits for client-requests, receives, buffers and sends data.

The communication is done based on sockets (BSD Sockets) and uses the TCP/IP protocol. Clients and server may run on different computers in the network. For the socket-connection, a port-number and the server-host-name must be specified. The ACI library contains the communication layer both for the client and server side.

Application layer - ACCI

The application layer, the ACCI module (AVL Code Coupling Interface), provides the functionality necessary for the coupling, which is data-transfer and buffering, synchronization of the processes and mapping attribute values between different meshes. It also reads the coupling-configuration file, which defines the values (physical quantities), the geometric locations and the clients that send or receive the data.

The ACCI module consists of the ACCI coupling server (program **acci_server**, included into the FIRE installation) and the ACCI client library.

24.2. Boost Link (Plenum)

Element Name:	**Boost Link (Plenum)**	
Element Icon:		
Definition:	This element defines the properties of an Interface between BOOST Hydsim Piston element and BOOST Plenum-type element.	
Connecting pins:	standard pins: 10 (hydraulic) special pins: 0 wire pins: 0	

Note: Currently the Variable-step solver does not support this element.

Note: **Boost Link (Plenum)** can be connected only to **Piston (Standard)** element (one or several).

Note: **Boost Link (Plenum)** element represents a volume type element filled with gas. Volume size of a plenum is a variable. BOOST Hydsim calculates plenum size change due to Piston motion and sends it to BOOST. BOOST calculates gas pressure in the plenum and sends it back to BOOST Hydsim.

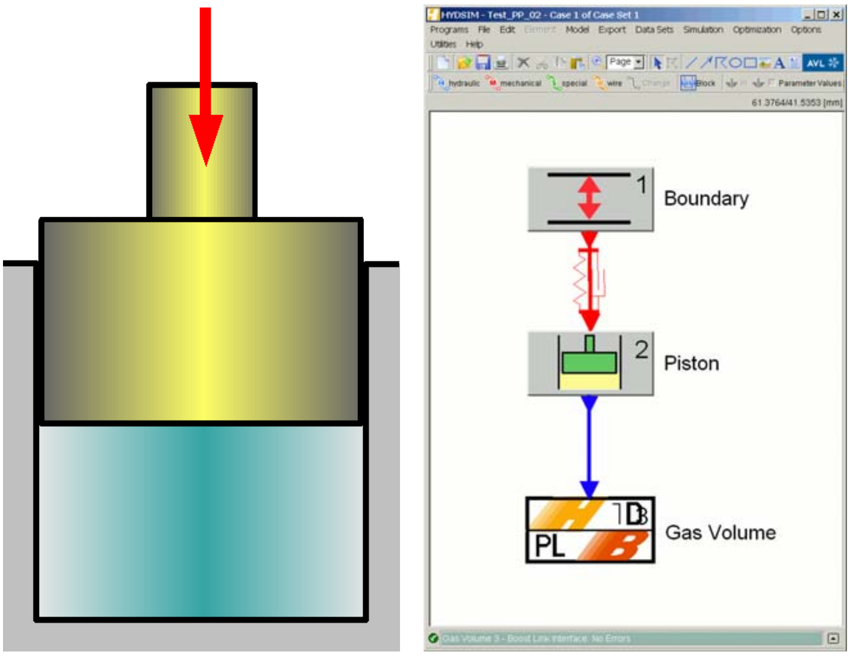


Figure 249: BOOST Hydsim model of simple Piston-Plenum system with Boost Link (Plenum) element

24.2.1. Input Parameters

Description of input data of **Boost Link (Plenum)** element.

Element name	The name of the element is specified as default.
GENERAL	
Channel Number	Define channel number for the communication (data exchange) with BOOST Plenum element. Number of channels has to be same as number of Boost (Link) elements in the model. Through one channel single BOOST Hydsim element exchanges data with single Boost element.

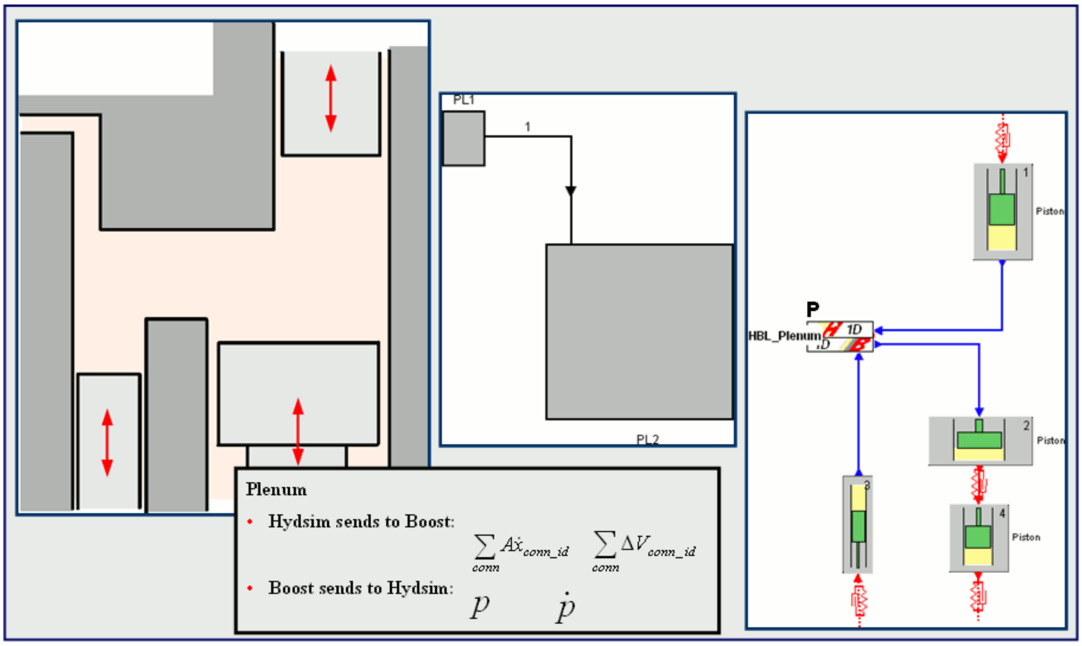


Figure 250: Communication between BOOST and BOOST Hydsim

Data output to files	
Write Boost data to file (optional)	BOOST data transferred to BOOST Hydsim via ACCI interface can be stored onto HBL_Boost_Data.dat file.
	List of variables which BOOST transfers to BOOST Hydsim: CHANNEL ················ Channel number through which data are exchanged SbC1 ························ Pressure in plenum (calculated in Boost) SbC2 ························ Pressure derivative in plenum (calculate in Boost)
Write BOOST Hydsim data to file (optional)	BOOST Hydsim data transferred to BOOST via ACCI can be stored into HBL_BOOST Hydsim_Data.dat file.
	List of variables which BOOST Hydsim transfers BOOST: CHANNEL ················ Channel number through which data are exchanged SbC1 ························ Sum of volumetric flow rates for all connected pistons (calculated in BOOST Hydsim) SbC2 ························ Sum of all piston displacements (calculated in BOOST Hydsim)

24.2.2. Initial Conditions

Path: **Element | Initials**

Initial conditions cannot be specified for **Boost Link (Plenum)** element.

24.2.3. Modifiable Parameters

Path: **Element | Modify**

Modifiable parameters cannot be specified for **Boost Link (Plenum)** element.

24.2.4. Output Parameters

Path: **Element | Store Results**

To activate output parameter, the option button on the left of output parameter name has to be checked.

List of output parameters:

static pressure at plenum
pressure derivative at plenum
sum of volumetric flow rates /time
sum of all displacements

At each exchange time Boost sends to BOOST Hydsim:

- pressure p and its time derivative $\dot{p}$ calculated in Plenum element of the Boost model. They are applied on all connected piston type elements. For each connected Piston, pressure p is multiplied with pressurized area for the calculation of hydraulic force.

At each exchange time BOOST Hydsim sends to Boost:

- sum of volumetric flow rates /time $\sum_i \dot{x}_i A_i$ and sum of all displacements $\sum_i x_i A_i$.

  Here i is the number of connected piston-type element to Boost Link (Plenum) element. It can be connected to up to 10 piston-type elements in BOOST Hydsim.

24.2.5. Data Exchange

Data transfer between BOOST Hydsim and BOOST is performed via ACCI interface. The interface program is launched by BOOST at calculation start. During initialization BOOST Hydsim and BOOST exchange control data like number of interfaces, type of interface (link to Plenum or Valve element in Boost), start and end of calculation in crank angle domain. This information is necessary for BOOST.

During the calculation data between BOOST Hydsim and BOOST are transferred in a vector of variable dimension. Size of the exchange vector is automatically adjusted according to the number of interfaces specified during initialization. At each data exchange time BOOST Hydsim provides to BOOST information about the next exchange time (or time step Δt till the next exchange event). It serves as a target time for BOOST. Data exchange between BOOST Hydsim and BOOST with local time steps is illustrated in Figure 251.

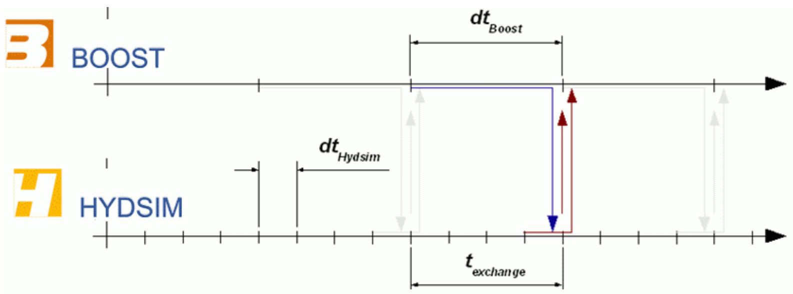


Figure 251: BOOST, BOOST Hydsim and exchange time steps

24.3. Boost Link (Valve)

Element Name:	**Boost Link (Valve)**	
Element Icon:		
Definition:	This element defines the properties of an Interface between BOOST Hydsim Piston element and BOOST Check Valve element.	
Connecting pins:	standard pins: 2 (hydraulic) special pins: 0 wire pins: 0	

Note: Currently the Variable-step solver does not support this element.

Note: **Boost Link (Valve)** can be connected only to **Piston (Standard)** element via hydraulic connection.

Note: Both hydraulic connections (upstream and downstream) of **Boost Link (Valve)** may be anchored to same Piston element (exception from standard BOOST Hydsim rule).

Note: Both connecton anchors of **Boost Link (Valve)** element may have same (input/input or output/output) direction (exception from standard BOOST Hydsim rule).

Note: **Boost Link (Valve)** has two hydraulic anchors: upstream UP & dowstream DW.
Area of Piston which is hydraulically connected to UP-anchor will be pressurised with upstream pressure from the Boost Valve element.
Area of Piston which is hydraulically connected to DW-anchor will be pressurised with the downstream pressure from the Boost Valve element.

24.3.1. Input Parameters

Description of input data for the **Boost Link (Valve)** element.

Element name	The name of the element is specified as default.
GENERAL	
Channel Number	Define channel number for the communication (data exchange) with BOOST Valve element. Number of channels has to be same as number of Boost (Link) elements in the model. Through one channel single BOOST Hydsim element exchanges data with single Boost element.
Data output to files	
Write Boost data to file (optional)	BOOST data transferred to BOOST Hydsim via ACCI interface can be stored into HBL_Boost_Data.dat file
	List of variables which BOOST transfers to BOOST Hydsim: CHANNEL .... Number of channel through which data are exchanged SbC1 .... Upstream valve pressure (calculated in Boost) SbC2 .... Upstream valve pressure derivative (calculate in Boost) SbC3 .... Downstream valve pressure (calculated in Boost) SbC4 .... Downstream valve pressure derivative (calc. in Boost)
Write BOOST Hydsim data to file (optional)	BOOST Hydsim data transferred to BOOST via ACCI interface can be stored into HBL_BOOST Hydsim_Data.dat file
	List of variables which BOOST Hydsim transfers BOOST: CHANNEL .... Number of channel through which data are exchanged SbC1 .... Valve body coordinate (calculated for piston element in BOOST Hydsim) SbC2 .... Valve body velocity (calculated for piston element in BOOST Hydsim)
VALVE BODY	
Position of valve body	
	x-coordinate of upper piston input end Piston is connected to UP-anchor. X-coordinate of piston input end will be transferred via Link element to Boost valve element.
	x-coordinate of upper piston output end Piston is connected to UP-anchor. X-coordinate of piston output end will be transferred via Link element to Boost valve element.
	x-coordinate of lower piston input end Piston is connected to DW-anchor. X-coordinate of piston input end will be transferred via Link element to Boost valve element.

	x-coordinate of lower piston output end Piston is connected to DW-anchor. X-coordinate of piston output end will be transferred via Link element to Boost valve element.

24.3.2. Initial Conditions

Path: **Element| Initials**

Initial conditions cannot be specified for **Boost Link (Valve)** element.

24.3.3. Modifiable Parameters

Path: **Element| Modify**

Modifiable parameters cannot be specified for **Boost Link (Valve)** element.

24.3.4. Output Parameters

Path: **Element| Store Results**

To activate output parameter, the option button on the left of output parameter name has to be checked.

List of output parameters:

static pressure before valve
pressure derivative before valve
static pressure after valve
pressure derivative after valve
valve body coordinate
valve body velocity

At each exchange time Boost sends to BOOST Hydsim:

- Pressure before valve p_{up}, its gradient $\dot{p}_{up}$, pressure after valve p_{dw}, its gradient $\dot{p}_{dw}$. Area of Piston element connected to UP-anchor will be pressurrized with pressure before Boost Valve element. Area of Piston element connected to DW-anchor will be pressurised with pressure after Boost Valve element.

At each exchange time BOOST Hydsim sends to Boost:

- valve body coordinate x_{vl} and velocity $\dot{x}_{vl}$.

24.3.5. Examples

Some characteristic gas valve examples modeled using BOOST-BOOST Hydsim link are shown in Figure 252, Figure 253, Figure 254, and Figure 255. BOOST Hydsim submodels include **Boost Link (Plenum)** and **Boost Link (Valve)** with one (UP-upstream) or two (UP-upstream and DW-downstream) connection achors. Each **Boost Link (Valve)** element in BOOST Hydsim corresponds to **Check Valve** element in the BOOST model. The exchange variables between BOOST Hydsim andf BOOST are shown on every plot.

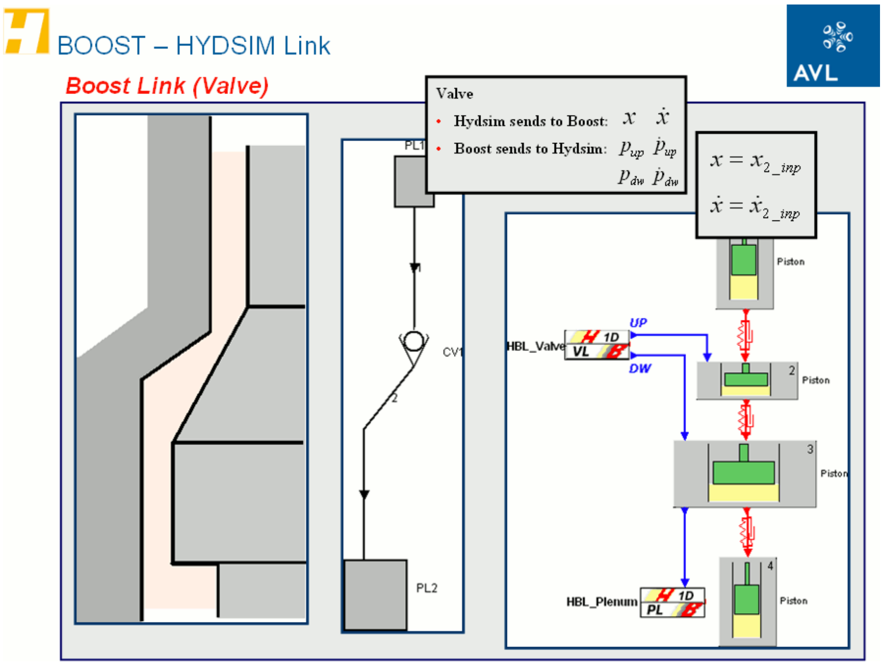


Figure 252: BOOST & BOOST Hydsim models of gas valve with conical seat (BOOST Link element in BOOST Hydsim with UP & DW anchors)

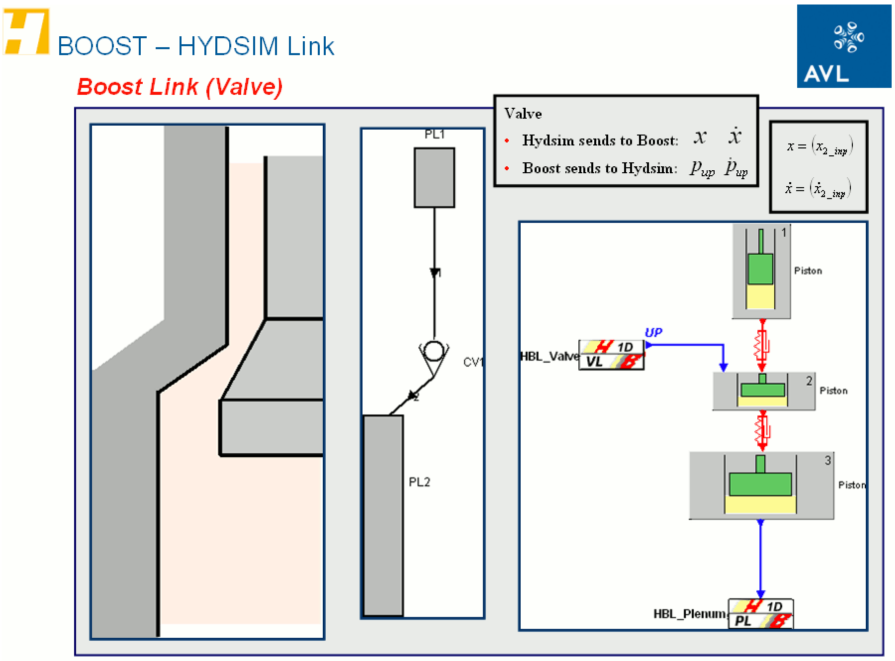


Figure 253: BOOST & BOOST Hydsim models of gas valve with conical seat (BOOST Link element in BOOST Hydsim with one UP anchor)

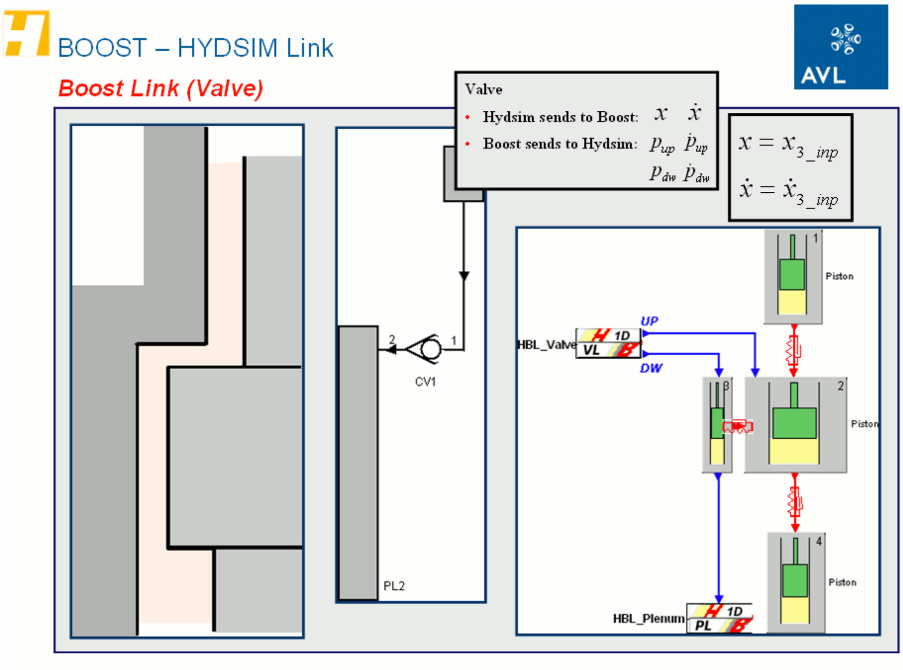


Figure 254: BOOST & BOOST Hydsim models of gas valve with flat seat (BOOST Link element in BOOST Hydsim with UP & DW anchors)

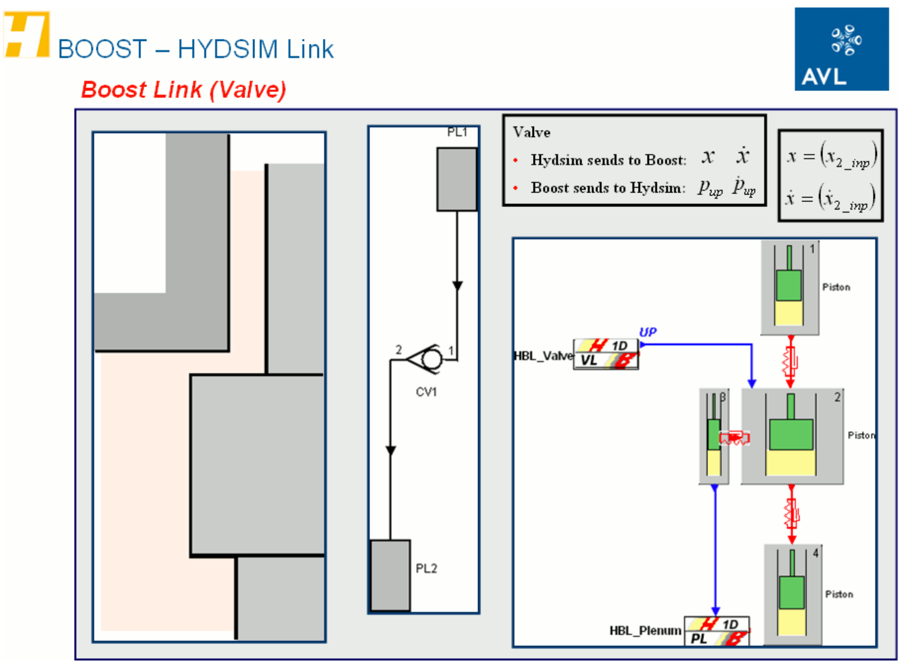


Figure 255: BOOST & BOOST Hydsim models of gas valve with flat seat (BOOST Link element in BOOST Hydsim with one UP anchor)

25. CONNECTIONS

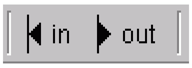
There are four types of element connections in BOOST Hydsim:

- **Mechanical** (red)
- **Hydraulic** (blue)
- **Special** (green)
- **Wire** (orange)

Red and blue arrows represent mechanical and hydraulic connections. According to arrowhead direction, BOOST Hydsim identifies whether the connection is on the input or output end of element.

If arrowhead is directed into element, the connection will be at input end of element (input connection for that element). If arrowhead is directed away from the element, the connection will be at output end of element (output connection for that element).

Each mechanical or hydraulic connection must have one input and one output end. Input-input and output-output connections are possible but should not be used (they usually are not necessary).

To change the direction of arrowhead, use buttons `in` or `out` 

on left-side palette. Select the arrowhead and click on the selected button to change.

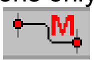
Green lines represent special (logical) connections and orange lines represent wire connections (to MATLAB elements). These connections have no direction.

25.1. Mechanical Connection

Connection Name:	**Mechanical Connection**
Connection Icon:	
Color:	Red
Definition:	This connection defines the mechanical preload, stiffness, and damping between two elements. Additionally, for the connected elements the pressure-active area as function of relative displacement can be defined.

Note: Multiple **Mechanical connections** between same two elements in same direction cannot be specified. All mechanical connections except the first will be ignored.

Note: **Mechanical connections** transfer only mechanical forces (from preload, stiffness, and damping) between the two connected elements. **Exception: Mechanical connection** will transfer information about pressure from **Hydromechanical boundary** to a piston-type element (**Piston**, **Plunger**, **Needle**, etc.).

Note: **Mechanical connections** can be established between **red** or **black** connection anchors of element icons only. To activate connection anchors and draw a **Mechanical connection**,  button has to be pressed.

Note: **Mechanical connections** can be defined in three directions: x (translation), y (translation), and w (rotation). The actual direction has to be specified in the **Input Dialog** box.

25.1.1. Input Data

The **Input Data Dialog Box** of **Mechanical Connection** is shown in Figure 256 (**Preload** Sheet active), (**Stiffness** Sheet active), (**Damping** Sheet active) and (**Area** Sheet active). Input of preload, stiffness, damping and area values is discussed in Sections 25.1.1.1, 25.1.1.2, 25.1.1.3 and 25.1.1.4, respectively.

25.1.1.1. Preload Input

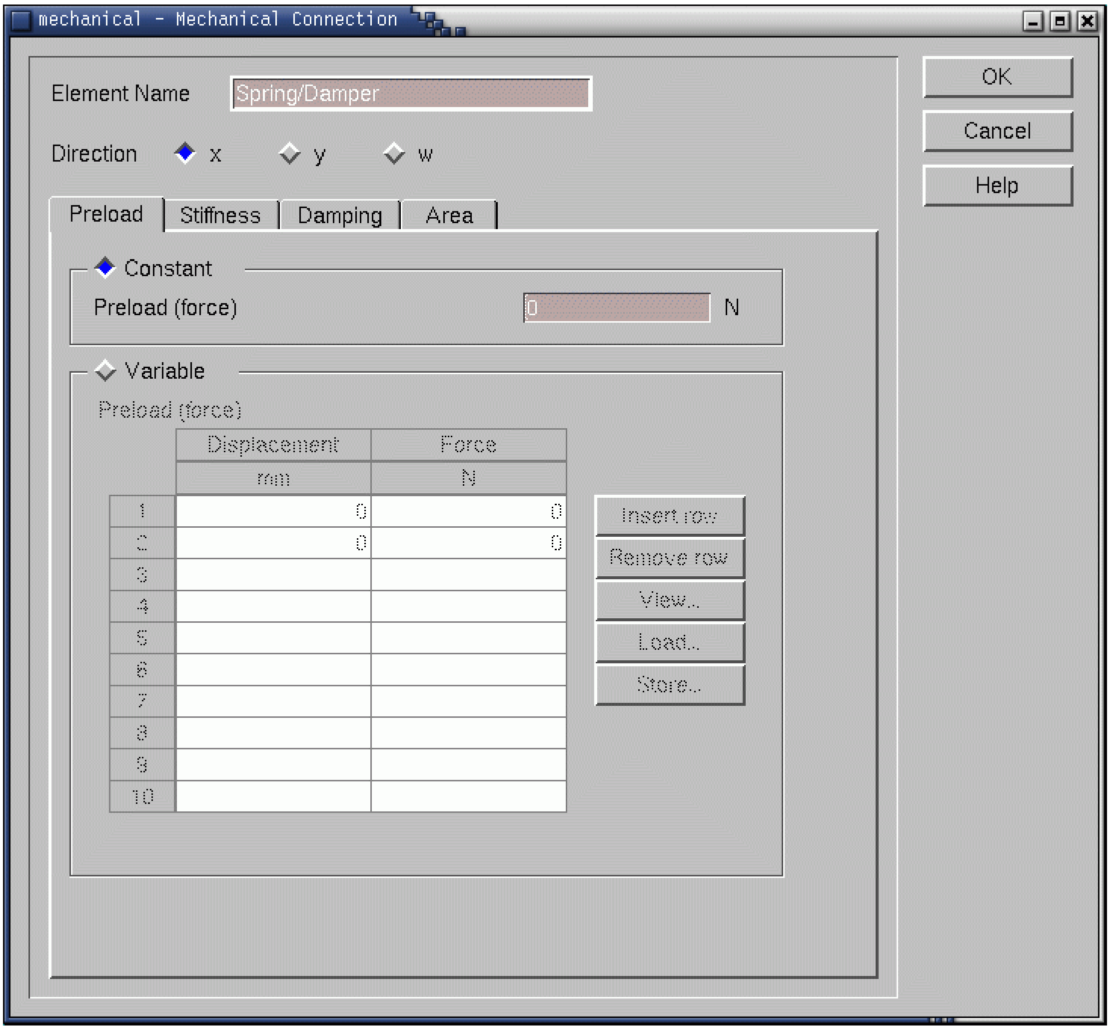


Figure 256: Preload Data Dialog Box of Mechanical Connection

Description of input data:

Element name	The name of the element is specified as default.
Direction	Specify in which direction the **Mechanical connection** acts. It may be x-direction (translation), y-direction (translation), and w-direction (rotation).
PRELOAD TAB	
Constant	Click to activate constant spring preload.
Preload (force/torque)	Specify the constant spring preload force or torque between two elements. This input is possible only if the **Constant** button is pressed.
Variable	Click to activate variable coordinate-dependent spring preload between two elements.

Preload (table)	Specify the spring preload as a function of relative coordinate (displacement or angle) between connected elements in form of table. The following rules apply to the preload table: 1. Displacement (angle) values in the 1st column must be given in ascending order. 2. Positive displacement (angle) values with "+" sign correspond to compression of spring. 3. Negative displacement (angle) values with "-" sign correspond to extension of spring. 4. Intermediate values are bridged by linear interpolation. 5. If the calculation exceeds the defined range, the first or last preload value is kept constant for the further calculation until the relative displacement (angle) returns to the table range.

☞ **Note:** Usually spring preload is constant, therefore the table of variable preload is seldom used. However, it may be useful in special cases (e.g. contact loss modeling).

25.1.1.2. Stiffness Input

STIFFNESS TAB	
Constant	Click to activate constant translational (rotational) stiffness.
Translational stiffness	Specify the constant linear or rotary stiffness between two elements. This input is active only if the **Constant** button is checked.
Variable	Click to activate variable (coordinate-dependent) stiffness between two elements.
Stiffness table	Specify the spring stiffness as a function of relative coordinate (displacement/angle) between connected elements in form of table. General rules for the specification of stiffness table: 1. Displacement (angle) values in the 1st column must be given in ascending order. 2. Positive displacement (angle) values with "+" sign correspond to compression stiffness of spring. 3. Negative displacement (angle) values with "-" sign correspond to tensile stiffness of spring. 4. Intermediate values are bridged by linear interpolation. 5. If the calculation exceeds the defined range, the first or last preload value is kept constant for the further calculation until the relative displacement (angle) returns to the table range. Refer to Section 25.1.1.1 for information on the data input into the table.

☞ **Note:** If the stiffness table contains only positive values in the first column, the spring stiffness characteristic is defined for spring compression only. For spring extension, BOOST Hydsim will assume zero stiffness (contact loss).

25.1.1.3. Damping Input

DAMPING TAB	
Constant	Click to activate constant translational (rotational) damping.
Translational damping	Specify the constant damping between two elements. This input is active only if the **Constant** button is checked.
Variable	Click to activate variable (coordinate or velocity dependent) damping between two elements.
Damping as function of Coordinate / Velocity	Click to activate damping as function of relative coordinate or velocity of connected elements.
Damping table	For **Damping as function of Coordinate:** specify the damping coefficient as a function of relative displacement (angle) between connected elements in the table. For **Damping as function of Velocity:** specify the damping coefficient as a function of relative linear (angular) velocity between connected elements in the table. General rules for the specification of damping table: For **Damping as function of coordinate:** 1. Displacement (angle) values in the 1^{st} column must be given in ascending order. 2. Positive displacement (angle) values with "+" sign correspond to compression damping of spring. 3. Negative displacement (angle) values with "-" sign correspond to tensile damping of spring.

	For **Damping as function of velocity**: 1. Velocity value in the first column must be given in ascending order. 2. Positive velocity (value with "+" or no sign) corresponds to compressive damping (connected elements approaching each other). 3. Negative velocity (value with "-" sign) corresponds to tensile damping (connected to elements moving apart from each other. 4. Intermediate values are bridged by linear interpolation. 5. If the calculation exceeds the defined range, the first or last damping value is kept constant for the further calculation until the relative coordinate (velocity) returns into the table range again.

☞ **Note:** If the damping table contains only positive values in the first column, the damper characteristic is defined for damper compression only. For damper extension, BOOST Hydsim will assume zero damping.

25.1.1.4. Area Input

AREA TAB	
Constant	Click to activate constant pressure-active area. The areas will be taken from the input dialogs of the connected elements.
Variable	Click to activate variable pressure-active area. **Note:** Area table will overwrite the respective area/diameter data in the input dialogs of the connected elements.
Area table	Specify the input and output area as a function of relative coordinate (displacement) between the connected elements. General rules for the specification of area table: 1. Displacement values in the 1^{st} column must be given in ascending order. 2. Positive displacement values with "+" sign imply the overlapping of connected elements by displacement value. 3. Negative displacement (angle) values with "-" sign imply the staying apart of connected elements at absolute displacement. 4. Intermediate values are bridged by linear interpolation. 5. If the calculation exceeds the defined range, the first or last area value is kept constant for the further calculation until the relative displacement (angle) returns to the table range. Refer to Section 25.1.1.1 for information on the data input into the table.

☞

Note: **Area sheet** is active only for x-direction (the only hydraulic direction).

Note: **Variable Area table** (column) is active only for the elements which have a hydraulic connection in parallel with mechanical connection. Both connections must have same direction (input or output).

Note: **Area table** (column) can be activated only for elements: **Standard Piston** (both sides), **Split-Injection Piston** (both sides but variable input area cannot be bigger than seat area), **Plunger** (both sides), **Solenoid Armature** (both sides), **Needle** (only output side). For **Hollow Needle** variable area input is disabled.

Note: Within a single element, **Variable Area table** can be activated only for one input and one output mechanical connection. For the other parallel mechanical connections (if any), **Variable Area table** is disabled.

Note: According to arrowhead direction, BOOST Hydsim identifies whether or not the connection is on the input or output end of element and overwrites input or output area/diameter data in connected elements with values from area table.

25.1.2. Variable Mechanical Connections

BOOST Hydsim allows variable damping, stiffness, and preload characteristics of connections for two translational directions (along the x and y axes) and one rotational direction (around w axis). Explanation for x direction follows.

Damping, stiffness and force (preload) can be non-linear functions of relative displacement between two connected elements. Furthermore, damping can be a non-linear function of the relative velocity:

$$\begin{aligned} c_{12} = c(\dot{x}_1 - \dot{x}_2) \quad &or \quad c_{12} = c(x_1 - x_2) \\ k_{12} &= k(x_1 - x_2) \\ F_{12} &= F(x_1 - x_2) \end{aligned} \tag{484}$$

where c_{12}, k_{12} and F_{12} are damping, stiffness, and preload force, respectively, and x_1 and x_2 are absolute displacements of the connected bodies (refer to Figure 257).

Obviously, the variable characteristics of the connection between the first and the second body depend on the coordinate and velocity of the connected elements.

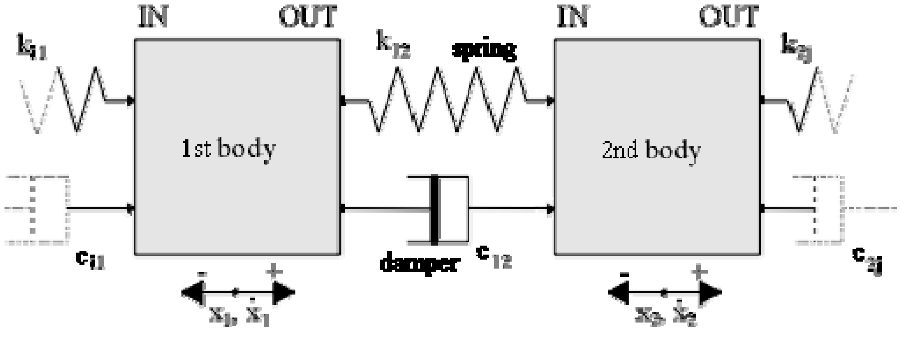


Figure 257: Mechanical Connections between Two Bodies with Input/Output Directions

Positive direction for coordinate and velocity always goes from input to output end of the body. Direction is important for dynamic calculation when we have other irreversible connections (connections to non-symmetric elements with different input and output ends).

Positive sign for the values in the first column of variable stiffness or force table implies $x_1 - x_2 > 0$ or the spring being compressed. Negative sign implies $x_1 - x_2 < 0$ or the spring being extended.

Similar situation is the with sign of the relative velocity between the first and the second body: positive sign „+“ implies $\dot{x}_1 - \dot{x}_2 > 0$ or damper being compressed, negative sign „-„ implies $\dot{x}_1 - \dot{x}_2 < 0$ or damper being extended.

25.1.3. Variable Area

Variable Area serves to define the variable **pressure-active area** on input or output side of a piston-type element. It can be specified as a function of relative displacement between the two piston-type elements or one piston-type element and another mechanical element (e.g. mechanical boundary).

With **variable damping, stiffness, and preload characteristics** it is possible to simulate the contact between two piston-type elements and adequate mechanical forces on them. With **variable pressure-active area** it is possible to model hydraulic forces which depend not only on the input and output pressure but also on the piston displacement. An example of such a case is provided in Figure 258. As long as Piston 1 and Piston 2 stay apart form each other ($x_{dist} > 0$), the pressure in the middle volume p_{mid} acts on the area A_{1_out} of Piston 1and area A_{2_inp} of Piston 2. However, as soon as the pistons come into contact and move like one rigid body, the pressure-active area on output side of Piston 1 vanishes to zero, while the pressure-active area on input side of Piston 2 reduces to $(A_{2\_inp} - A_{1\_out})$.

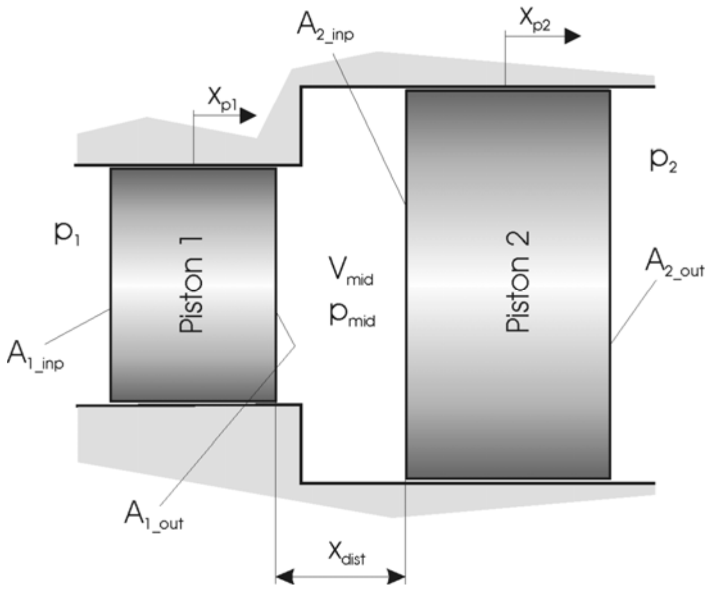


Figure 258: Variable Pressure-active Area between Two Pistons

Hydraulic force F_{hyd} acting on Piston 1 is calculated from the equation:

$$F_{hyd\_p1} = p_1 A_{1\_inp} - p_{mid} A_{1act\_out} \tag{485}$$

where A_{1act_out} is the pressure-active output area of Piston 1 given by:

$$\begin{cases} \forall x_{p1} - x_{p2} < 0 & A_{1act\_out} = A_{1\_out}, \\ \forall x_{p1} - x_{p2} \geq 0 & A_{1act\_out} = 0. \end{cases} \tag{486}$$

Hydraulic force F_{hyd} acting on Piston 2 is calculated from the formula:

$$F_{hyd\_p2} = p_{mid} A_{2act\_inp} - p_2 A_{2\_out}, \tag{487}$$

where A_{2act_inp} is the pressure-active input area of Piston 2 given by:

$$\begin{cases} \forall x_{p1} - x_{p2} < 0 & A_{p2\_inp} = A_{2\_inp}, \\ \forall x_{p1} - x_{p2} \geq 0 & A_{2act\_inp} = A_{2\_inp} - A_{1\_out}. \end{cases} \tag{488}$$

The above relationships between the areas and relative displacement have to be defined in the input dialog of **Variable Area**.

25.2. Hydraulic Connection

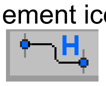
Connection Name:	**Hydraulic Connection**
Connection Icon:	
Color:	Blue
Definition:	This connection enables pressure and/or flow rate exchange between connected elements.

Note: x direction is the only hydraulic direction. Multiple **Hydraulic connections** between same two elements cannot be specified.

Note: **Hydraulic connection** transfers pressure or flow rate from one element to the other. In this way, an element which expects flow rate from the connection and delivers pressure to the connection (this is **Volume** element only) has to be connected to the element which delivers flow rate to the connection and expects pressure from it (e.g. **Orifice**, **Line**, **Throttle**). Hence connecting e.g. two **Volume** elements is not possible and will produce an error message. However, connections **Line-Line**, **Line** -**Orifice** etc. are formally allowed in GUI. In this case BOOST Hydsim inserts a small virtual **Volume** of relevant type in between to satisfy the boundary conditions.

Note: If a piston-type element is connected with a **Volume** element, the displacement of **Piston** will generate volumetric flow rate into the **Volume**.

Note: **Hydraulic connection** is only an arrow pointing to the possible flow direction. It should not be mixed with a **Line** element. Hence there is no input data associated with a hydraulic connection.

Note: **Hydraulic connections** can be established only between **blue** connection anchors of element icons. To activate connection anchors and draw a **Hydraulic connection**, 

button has to be pressed.

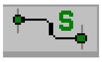
25.3. Special Connection

Connection Name:	Special Connection
Connection Icon:	
Color:	Green
Definition:	Special connection enables position- or velocity-controlled elements (Spill Port, Leakage, Lift Controlled Throttle, etc.) to receive/send information about coordinate and velocity from/to connected controlling elements (piston-type elements, valve bodies, etc.).

Note: Information about coordinate or velocity is transferred by **Special connection** in only one direction (e.g.: from **Plunger** to **Spill Port**, from **Piston** to **Leakage**, etc.).

Note: **Special connection** is essentially a logical connection. It is necessary for the „on-line" data exchange between two elements belonging to one physical unit or control device.

Note: There is no Input Data associated with a **Special connection**.

Note: Special connections can be established only between **green** connection anchors (semicircles) of element icons. To activate connection anchors and draw a **Special connection**,  button has to be pressed.

25.4. Wire Connection

Connection Name:	**Wire Connection**
Connection Icon:	
Color:	Orange
Definition:	Wire connection enables Control elements (Simulink, m-function, DLL and PID Controller) to exchange input/output parameters with other BOOST Hydsim elements.

Note: Input/output parameters are transferred by **Wire connection** in both directions (from **BOOST Hydsim element** to **MATLAB® element** and backwards).

Note: There is no Input Data associated with a **Wire connection**.

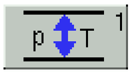
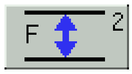
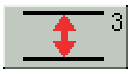
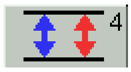
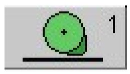
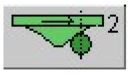
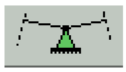
Note: **Wire connection** can be established only between **orange** connection anchors of element icons. To activate connection anchors and draw a **Wire connection**,  button has to be pressed.

26. GRAPHICAL USER INTERFACE (GUI) AND COMPONENTS

26.1. Element Library

Element library in BOOST Hydsim consists of various hydraulic, mechanical and control elements. Elements are collected into groups according to their type and functionality. BOOST Hydsim has nineteen element groups. Group and element names are listed in the element tree on the left side of GUI main window.

Table 12: Element Library

GROUP	ELEMENT	ICON
Boundary	**Pressure/Temperature** This element serves to define the pressure and temperature on outside connections (boundaries) of the system as function of time or reference angle.	
	Flow Rate This element serves to define the flow rate on outside connections (boundaries) of the system as function of time or reference angle.	
	Mechanical This element serves to define the coordinate or velocity on outside connections (boundaries) of the system as function of time or reference angle.	
	Hydromechanical This element serves to define 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** This element serves to define a cam profile by the follower acceleration or lift data (grinding coordinates for reciprocating follower).	
	Cam Plate * This element serves to define a cam plate of distributor-type injection pump (e.g. Bosch VE30) by acceleration of lift data.	
Lever	**Rocker Arm (Pushrod)** * This element serves to define a pushrod-actuated rocker arm.	

	Rocker Arm (Cam) * This element serves to define a cam-actuated rocker arm. It includes cam profile in polar coordinates.	
	Finger Follower * This element serves to define a finger follower (semi-floating rocker). It includes cam profile in polar coordinates.	
Solid	**Lumped Mass** This element serves to define a lumped mass (with two DOFs).	1
	Rigid Shaft This element serves to define a shaft (with three DOFs).	2
Piston	**Standard** This element serves to define a standard (hydraulic) piston.	1
	Split-Injection (SID) * This element serves to define a split-injection-device piston. SID piston is used for rate shaping of fuel injection (e.g. in pilot injection systems).	2
Volume	**Standard** This element serves to define a standard volume with rigid walls.	1
	Compliant This dialog serves to define a volume with compliant walls (cylindrical or spherical).	2
	Two-phase * This element serves to define a two-phase volume (with cavitation or fluid-gas mixture).	3
Line	**d'Alembert Model** This element serves to define a line (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.	1
	Laplace Transform This element serves to define a (tube/pipe) line of circular form. The solution of the line equation is obtained by Laplace transform (Kroller[41]). The non-stationary friction losses are calculated by Melcher's method.	2

[41] 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.

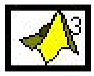
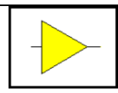
	Characteristics Method * This element serves to define a line (tube/pipe) solved by the method of characteristics. For integration, the Predictor - Corrector scheme is used. The non-stationary frictional losses are calculated by Melcher's method.	
	Godunov Method * This element serves to define a line (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) This element serves to define a line (tube/pipe) solved by MacCormack finite difference scheme[42]. 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** * This element serves to define a round bend of circular cross-section and constant area.	
	Mitre/Circular * This element serves to define a mitre bend of circular cross-section and constant area.	
Junction	**Tee (90 deg)** * This element serves to model Tee Junction with 90-deg angle between Side branch (Port 1) and Straight passage (Port 2 – 3).	
	Tee (angle) * This element serves to model Tee Junction with arbitrary angle between Side branch (Port 1) and Straight passage (Port 2 – 3).	
Pump	**Radial Piston (RPD)** * This element serves to define a radial piston distributor (RPD) pump. The model can account for the elastic Hertz contact between the pump elements.	
	Plunger This element serves to define the plunger (of any type).	
	Ejector (Hydraulic) This element serves to model the device in which a high-pressure fluid jet is used to drive low-pressure fluid.	

[42] 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** This element serves to model the fluid leakage through the annular gap between piston-type element (Piston, Plunger, Needle guide, etc.) and barrel (cylinder wall).	1
Port	**In-line Fill/Spill** This element serves to define a round filling, spill, or fill/spill port of in-line pump.	1
	Distributor Fill/Spill * This element serves to define the filling/spill ports of distributor-type rotary pump (e.g. Bosch VE).	2
Valve	**Delivery** This element serves to define a delivery valve. It can be used standalone or as a part of snubber valve or constant pressure valve.	1
	Constant Volume * This element serves to define a constant volume delivery valve with a retraction piston.	2
	Check (Ball) This element serves to define a pressure check valve with spherical body (ball).	3
	Check (Poppet) This element serves to define a pressure check valve with conical body.	4
Throttle	**Time-controlled (Switch)** This element serves to define a switch valve controlled by the timing of opening/closing areas (flow area as function of time or crank angle).	1
	Lift-controlled (Slide) This element serves to define a slide (spool) valve controlled by position of a mechanical body (flow area as function of position).	1
	Flow Area vs. Time/CA This element serves to define a throttle controlled by flow area variation vs. time (area as function of time or crank angle).	3
	Pressure Drop vs. Flow * This element serves to define a throttle controlled by pressure-flow diagram (pressure difference as function of flow rate).	ΔP Q 4
Solenoid	**Armature (Basic model)** This element serves to define a solenoid armature actuated by a gap-dependent magnetic force.	1

	Armature (Extended model) This element serves to define 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 *** This element serves to define a lift function of a piezoelectric actuator. It is a boundary condition with a predefined charging/discharging function.	
	Lift Amplifier * This element serves to define 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 * This element serves to define a piezoelectric stack actuator. It is a lumped-parameter model with rate-independent hysteresis describing the accurate actuator behavior in electrical and mechanical domains.	
Orifice	**General** This element serves to define a general orifice. It is not restricted to any specific geometry. Flow resistance coefficient has to be defined by the user.	
	Cavitating This element serves to define a cavitating orifice (flow model with cavitation). It is not restricted to any specific geometry.	
	Sharp-edged * This element serves to define a sharp-edged orifice. Flow resistance coefficient is calculated internally.	
	Round-edged * This element serves to define a round-edged orifice. Flow resistance coefficient is calculated internally.	
	Sharp-edged Long * This element serves to define a sharp-edged long orifice. Flow resistance coefficient is calculated internally.	

Nozzle	**SAC (Basic model)** This element serves to define a sac nozzle orifice with hydraulic flow (basic model).	1
	VCO (Basic model) This element serves to define a valve-covered nozzle orifice with hydraulic flow (basic model).	2
	SAC (Extended model) This element serves to define a sac nozzle orifice (extended model with cavitation).	3
	VCO (Extended model) This element serves to define a valve-covered nozzle orifice (extended model with cavitation).	4
	RSN-Collar Throttle * This element serves to define a needle collar throttle of rate-shaping nozzle (RSN).	5
	Non-circular Guide Flow * This element serves to define the properties of flow through needle guide: prismatic guide with segments.	6
	Standard This element serves to define a standard needle model of (single-spring) nozzle.	3
	2-spring This element serves to define a two-spring needle model of Injector (TSI) nozzle.	4
G **Gas**	**Pressure/Temperature *** This element serves to define the gas pressure and temperature on outside connections (boundaries) of the system as function of time or reference angle.	p T 1
	Gas Volume * This element serves to define a gas volume.	2
	Lift-controlled * This element serves to define a gas valve controlled by position of a mechanical body (flow area as function of position).	3
	Flow Area vs. Time/CA * This element serves to define a gas throttle controlled by flow area variation vs. time (area as function of time or crank angle).	4
C **Control**	**MATLAB® API: Simulink *** This element serves to define an interface between BOOST Hydsim and MATLAB® API Simulink.	1
	MATLAB® API: m-function * This element serves to define an interface between BOOST Hydsim and MATLAB® API m-function.	2

	MATLAB® DLL * This element serves to define an interface between BOOST Hydsim and MATLAB® DLL (dynamically linked library).	
	PID Controller * Defines Proportional, Integral and Derivative parameters (gains) of PID Controller.	
E **External**	**Fire Link (Nozzle)** * Defines an interface between BOOST Hydsim and FIRE nozzle flow co-simulation.	
	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.	

* Currently the Variable-step solver does not support these elements.

26.2. Control Points /General Settings

This chapter describes the settings in the top pull-down menu of BOOST Hydsim as follows:

- File (section 26.2.1)
- Edit (section 26.2.2)
- Element (section 26.2.3)
- Model (section 26.2.3.1)
- Export (section 26.2.5)
- Simulation (section 26.2.4.4)
- Utilities (section 26.2.7)

26.2.1. File

The File pull-down menu contains the following options:

New:	Opens new instance of BOOST Hydsim
Open:	Opens existing BOOST Hydsim model file (*.hyd)
Save:	Saves current BOOST Hydsim model file
Save As:	Saves current model file under user-defined name
Page Setup:	**Size** serves to customize the size and page orientation **Grid** serves to customize the grid in main GUI window **Unit** serves to set the default drawing area unit
Print:	Opens **Print** dialog box of the operating system

Print to File:	Opens **Print to File** dialog box for saving screen content on file in different graphic formats.
Exit BOOST Hydsim:	Quits BOOST Hydsim session Listing of most recently opened BOOST Hydsim model files

26.2.2. Edit

The Edit pull-down menu contains the following options:

Cut	
Copy	
Paste	
Delete	
Initial Conditions:	Together with **Select,** allows automatic definition of initial conditions for multiple elements. Mechanical initial conditions: **Cam Profiles** **Shafts** **Mass** **Cam Plate, Rocker Arm (Pushrod)** **Rocker Arm (Cam)** **Finger Follower** **Piston SID, , Valve, Solenoid** **Piston Standard, Plunger, Needle** **RPD Pumps** Hydraulic initial conditions: **Volume** **Line**
Modules:	Together with **Select,** allows to save a group of elements as one module. Each element in the group maintains its original properties. Saving group of elements as module is useful when same group of elements with their properties has to be loaded into another BOOST Hydsim model. **Load Module** **Save Module** **Save As User Module** **Save As Site Module** **Save As System Module**
Blocks:	Blocks are container objects consisting of elements and lines. The elements and lines in the block cannot be destroyed and are not accessible for graphical interactions such as moving or sizing. This makes a block useful for grouping the elements. By the same token a block can be used as a fully reusable component storing the connection information. Basic block actions: **Create** **Break Up**

EPU Macro List:	It is possible to **Add** a new macroelement. By default this will be empty (without contents). It is possible to **Remove** existing macroelement from list. In all elements belonging to this macro element, the Link to Excite button will be deactivated. It is possible to **Open** a macro element and change the name in the dialog.
Select:	It is possible to select all objects in the active page. It is also possible to select all elements, all connections, ... **All Objects** **All Elements** **All Connections** **All Decorations** **User Defined**
Order:	**Raise** **Lower**
Group	**Group** **Ungroup**

26.2.3. Element

The Element pull-down menu contains the following options:

Properties:	Defines element input parameters. Refer to the input dialog of the specific element.
Initial Conditions:	Defines initial values. Refer to the initial condition dialog of the specific element.
Store Results:	Defines the output results to be stored for a specific element. Refer to the output parameters dialog of the element.
Modify:	Defines the modifiable input parameter of a specific element (Section 26.2.3.2)
Copying:	Copies the data between two elements of the same type.

26.2.3.1. Store Results

This dialog serves to define results which should be stored for a specific element. Refer to specific elements for their respective output dialog. User has to choose the Output Values to be stored by selecting the corresponding box.

26.2.3.2. Modifiable Parameters

Parameter Variation vs. Time/Reference Angle

The **Modify Parameter** dialog (Figure 259) has the same structure for every element. To activate modification, the **Modify…** button has to be pressed. When modification is activated, parameter variation table has to be defined. Open it by pressing **Modify...** button.

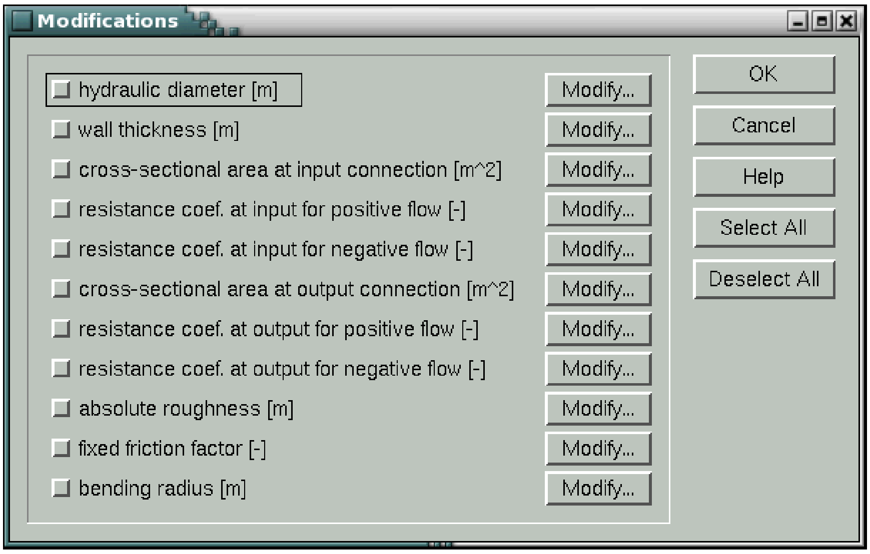


Figure 259: Modify Parameter (for Line element) Dialog

Pressing the **Modify** button on the **Modifications** screen yields the Input Data Dialog Box. This can be specified as a function of time or reference angle.

Description of input data:

Shift Time/Angle relative to calculation start	Specify the Shift time/angle for the first table column. It will be added to the time/angle values in the table at calculation start.
Scaling factor for 1st column	This button activates scaling factor for 1st table column. Values in the first column will be multiplied with the scaling factor as soon as **Run / Run Sets / Restart** is pressed.
TABLE	Specify time/ref. angle in ascending order in the first column. Specify the modified parameter in the second column. General rules for the specification: • Intermediate values are bridged by linear interpolation. • If the actual time/ref. angle exceeds the defined range, the last specified parameter value from the table is kept constant for the further calculation. • If the defined time/reference angle range for the modifiable parameter starts later than the calculation, the first specified parameter value is used.

26.2.4. Model

The Model pull-down menu contains the following options:

Parameters:	Definition of model parameters for Case Explorer. Refer to Section 26.2.4.1.
Case Explorer:	Definition of cases for the parameter variation with global and local variables etc. Refer to Section 26.2.4.3.

Fluid Properties:	Definition of global fluid properties in hydraulic system. Fluid properties can be constant or variable (pressure and/or temperature dependent). They can be defined in the dialog or taken from Fluid Property Database. Refer to Section 26.2.4.4.
Gas Properties:	Definition of global gas properties. Gas properties can be variable (basically temperature-dependent). They can be defined in the dialog or taken from Gas Property Database. Refer to Section 26.2.4.5.
Cylinder Charge:	Definition of global cylinder charge properties. Cylinder charge properties can be variable (basically pressure-dependent). They can be defined in the dialog or taken from Gas Properties. Refer to Section 26.2.4.6.
Solid Properties:	Definition of global solid properties. Solid properties can be constant or taken from Solid Property Database. If "Elastic body" option is not activated in any mechanical element, solid properties do not need to be defined. Refer to Section 26.2.4.7.
Property Database:	Definition of property database for fluids, gases and solids. Property Database can be accessed from every relevant element. Refer to Section 26.2.4.8.

26.2.4.1. Parameters

The **Parameter** dialog serves to define the parameters for the entire model. Click the menu item to open the following dialog:

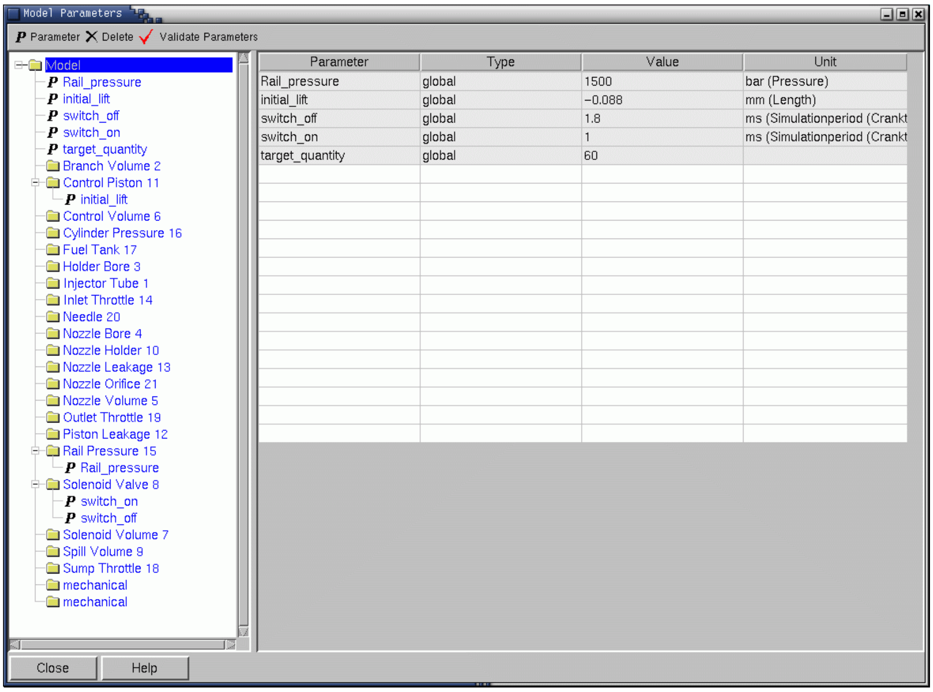


Figure 260: Model Parameter Dialog

This dialog displays the element tree with the created elements and defined parameters (variables). In the main window the specifications for the selected parameters can be entered.

To introduce a new parameter, highlight the **Model** item for global parameters or a specific **element** for local parameters. Then press the **Parameter** button on the top left corner.

26.2.4.2. Assigning Variables

To assign a variable to the specific element input parameter, place the mouse arrow on the text string of an input parameter and press the right mouse button. The following menu will pop up:

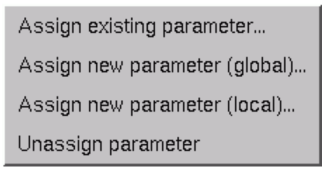


Figure 261: Assign Variable Menu

Select the menu item **Assign existing parameter** to open the following input dialog:

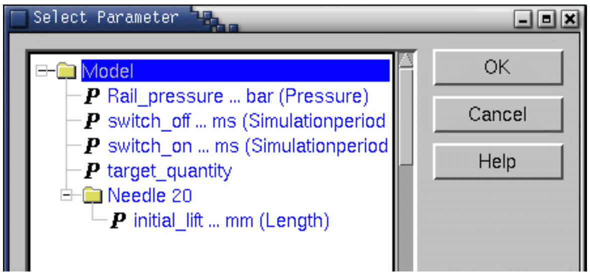


Figure 262: Select Parameter Dialog

To create a new parameter, click the menu item **Assign new parameter**.

26.2.4.3. Case Explorer

The Case Explorer enables the parameter value definition and controls the parameter variation.

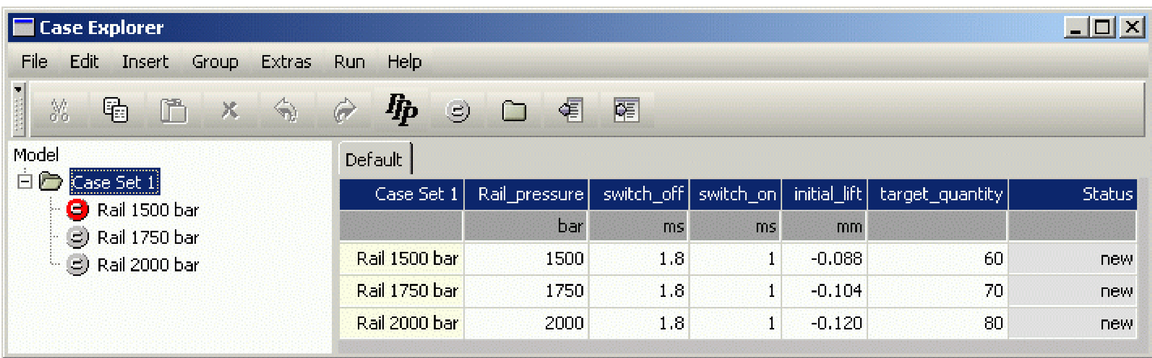


Figure 263: Case Explorer Dialog

The **Case Tree** with the defined simulation variants is displayed on the left side. The parameter values for each case table can be entered in the case table.

To define a new calculation case, select **Insert | Case Set** or click on .

To select cases for calculation, press the **Run…** button and select the desired variants in the list of **Cases** in the **Case Run** dialog. Click **Run** to start the simulation with all selected cases.

For further information please refer to the GUI Users Guide.

26.2.4.4. Fluid Properties

This dialog serves to define the global fluid properties in hydraulic system. Fluid properties can be **constant** or **variable**. If the system is purely mechanical (rare case), fluid properties do not need to be defined.

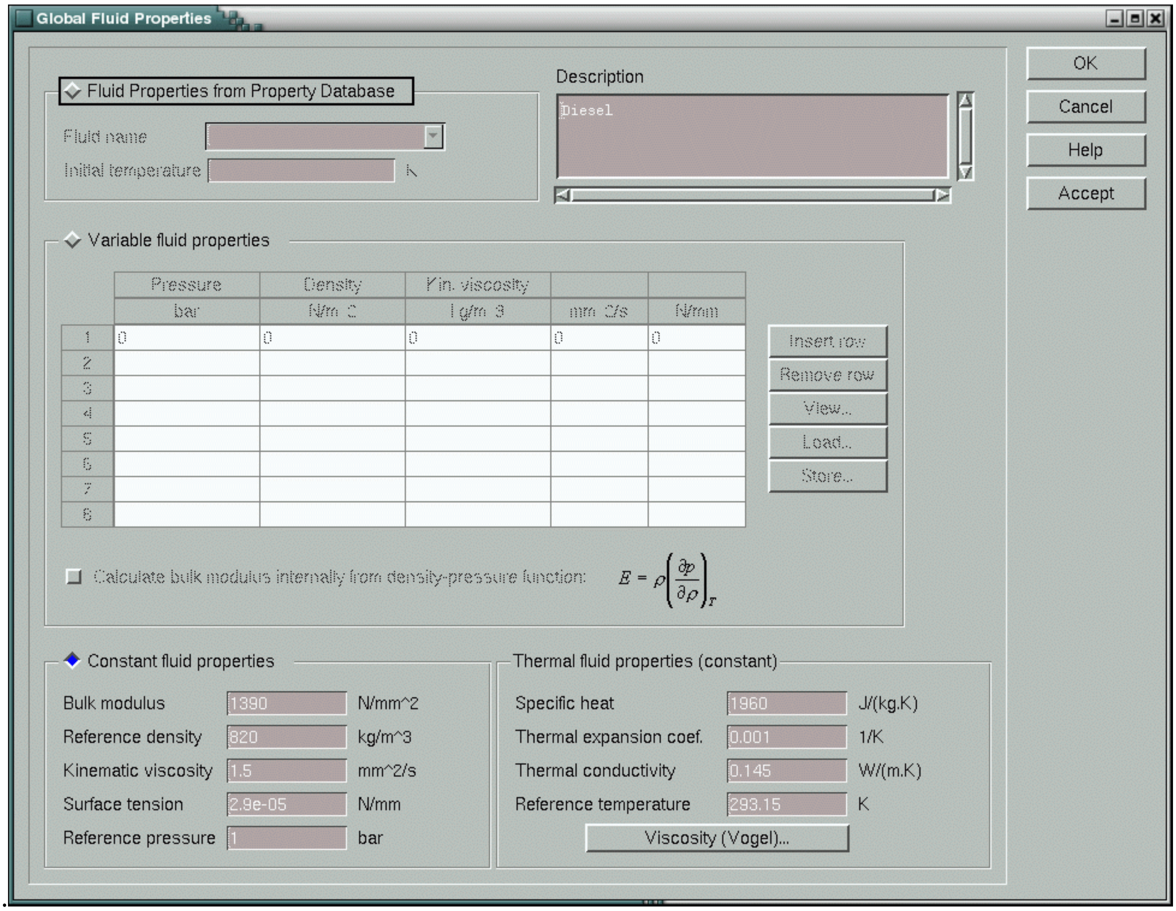


Figure 264: Global Fluid Properties Dialog

Description of input data:

FLUID PROPERTIES FROM PROPERTY DATABASE	Click to activate selection of fluid properties from Property Database where they are defined for each fluid from the list (refer to Chapter 26.2.4.8 for more information).
Fluid name	Specify fluid from the list.
Initial temperature	Specify either initial temperature of fluid for thermal calculation or constant fluid temperature for isothermal calculation.
VARIABLE FLUID PROPERTIES	Click to activate variable fluid properties in the system according to table data.

<table>
<tr>
<td>Table</td>
<td>
Specify the bulk modulus, reference density, kinematic viscosity and surface tension of the fluid as functions of pressure.
General rules for the specification of variable fluid properties:

Pressure values (first column) have to be specified in ascending order.
Intermediate values are bridged by linear interpolation. No extrapolation is performed.
If the pressure drops below the lowest value in the table (1st row), the values of fluid properties are taken from the 1st row and kept constant until the pressure reaches the defined range again.
If the pressure exceeds the highest values in the table (last row), then the values of fluid properties are taken from the last row and kept constant until the pressure drops down to the defined region again.

The pressure functions of bulk modulus, density and kinematic viscosity can be viewed in Impress Chart window by pressing View bar.
Density and Dynamic viscosity are considered as temperature dependent properties, while table values specify pressure dependence.
Density is calculated as:
$$\rho(T) = \rho_{ref} \cdot e^{-\alpha(T - T_{ref})},$$
where
ρ_{ref} reference density that is specified in table
α thermal expansion coefficient
(refer to Thermal fluid properties)
T fluid temperature
T_{ref} reference temperature coefficient
(refer to Thermal fluid properties)
Dynamic viscosity is calculated from Vogel viscosity formula:
$$\eta(T) = A \cdot e^{\frac{B}{T+C}} , \qquad (489)$$
where:
A reference dynamic viscosity $(= \nu \cdot \rho_{ref})$
ν specified Kinematic viscosity
B constant (refer to Vogel viscosity formula)
T fluid temperature
C constant (refer to Vogel viscosity formula)
</td>
</tr>
</table>

Calculate bulk modulus internally from density-pressure function	Click to activate internal calculation of Bulk modulus. In this case, Bulk modulus column will be grayed out. At least three rows have to be specified. Bulk modulus for each particular specified pressure is calculated from appropriate density and the partial derivation of the density at constant temperature with respect to pressure: $$E = \rho \frac{1}{\left(\frac{\partial \rho}{\partial p}\right)_T}$$
CONSTANT FLUID PROPERTIES	Click to activate constant fluid properties in the system.
Bulk Modulus	Specify the bulk modulus of fluid as global fluid property. It is kept constant through the whole calculation.
Reference density	Specify the fluid density at **Reference pressure and Reference temperature**. Density is considered as pressure and temperature dependent property and it is calculated as: $$\rho(p,T) = \rho_{ref} \cdot e^{\frac{1}{E}(p-p_{ref})-\alpha(T-T_{ref})},$$ where ρ_{ref} reference density E bulk modulus p fluid pressure p_{ref} reference pressure α thermal expansion coefficient (refer to **Thermal fluid properties**) T fluid temperature T_{ref} reference temperature coefficient (refer to **Thermal fluid properties**)
Kinematic viscosity	Specify the kinematics viscosity of fluid at reference pressure and reference temperature. It is used for calculation of reference Dynamic viscosity that is used in Vogel viscosity formula (eq. 489).
Surface tension	Specify the surface tension of fluid as global fluid property. It is kept constant through the whole calculation.
Reference pressure	Specify reference pressure at which are specified bulk modulus and reference density.
Velocity of sound is calculated according to formula: $$a = \sqrt{\frac{E}{\rho}},$$ where: a sound velocity in fluid E bulk modulus ρ fluid density	

THERMAL FLUID PROPERTIES (CONSTANT)	Active if either **Variable fluid properties** or **Constant fluid properties** is selected.
Specific heat	Specify specific heat of fluid. It is kept constant through the whole calculation.
Thermal expansion coef.	Specify thermal expansion coefficient of fluid. It is kept constant through the whole calculation.
Thermal conductivity	Specify thermal conductivity of fluid. It is kept constant through the whole calculation.

Reference temperature	Specify reference temperature at which bulk modulus, reference density and kinematic viscosity are specified.

☞ **Note:** If isothermal calculation task (under **Simulation | Mode** dialog box) is selected, only Reference temperature is required as input parameter, otherwise all four parameters are required.

VISCOSITY (VOGEL)	Press to specify constants *B* and *C* from Vogel formula (eq. 489). Parameter *A* from Vogel formula is equal to the Dynamic viscosity from the Fluid (Variable or Constant) properties: $\eta = \nu \cdot \rho$, where: η dynamic viscosity ν kinematic viscosity ρ fluid density
Constant B	Specify numerator from Vogel formula (eq. 489).
Constant C	Specify constant in denumerator from Vogel formula (eq. 489).

26.2.4.4.1. Density and Velocity of Sound

An experimental relationship between pressure and density of a typical diesel fuel is shown in Figure 264 [43]. The adequate relationship between the local pressure and velocity of sound is depicted in Figure 265. This figure shows that, over the operating pressure range of the fuel injection system, velocity of sound in the diesel fuel significantly depends on the pressure. For highly-compressible systems, like most fuel injection systems, this phenomenon cannot be neglected, i.e. variable fluid properties have to be considered.

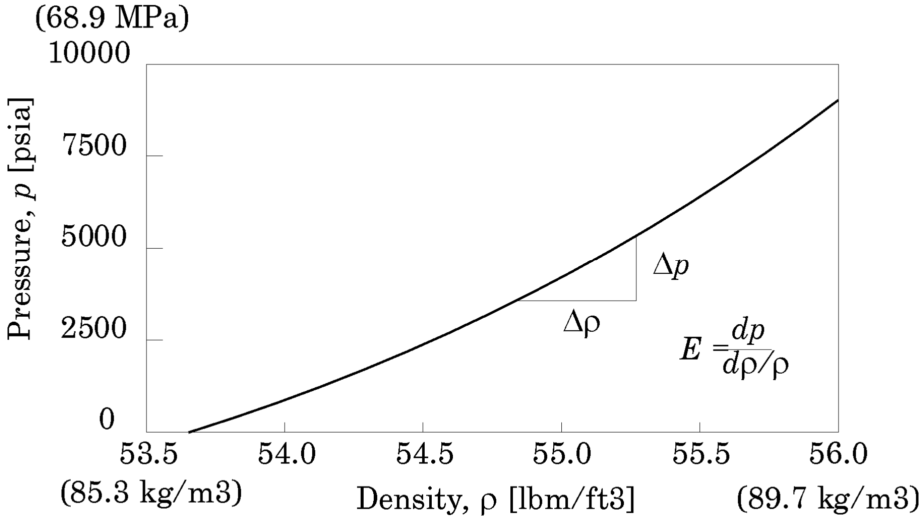


Figure 265: Compressibility Characteristics of Diesel Fuel

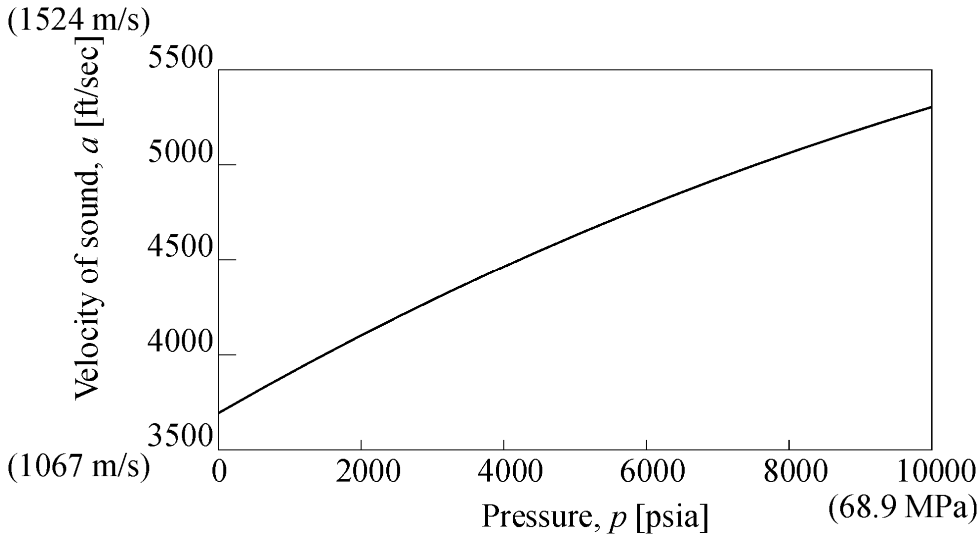


Figure 266: Velocity of Sound in Diesel Fuel

[43] Wright, W. *Prediction of the Bulk Moduli and Pressure-Volume-Temperature Data for Petroleum O*ils. Transactions of American Society of Lubrication Engineers, 10, 1967.

26.2.4.5. Gas Properties

Within **Global Gas Properties** dialog, the user may choose between two options:

1. **Gas properties from Property Database**
   Gas properties are chosen from the menu in Property Database (refer to Chapter 26.2.4.8 for more information)
   - Gas name [name of specie]
2. **Variable gas properties.**
   The tabular values of gas properties as a function of temperature have to be specified for the following properties:
   - Specific heat c_p [Gas Constant Units]
   - Specific heat ratio κ [Ratio Units]
   - Dynamic viscosity η [Dynamic Viscosity Units]

   General rules for the specification of gas properties:

   → Temperature values (first column) have to be specified in ascending order.

   → Intermediate values are bridged by linear interpolation. No extrapolation is performed.

   → If the temperature drops below the lowest value in the table (1st row), the values of gas properties are taken from the 1st row and kept constant until the temperature reaches the defined range again.

   → If the temperature exceeds the highest values in the table (last row), then the values of gas properties are taken from the last row and kept constant until the temperature drops down to the defined region again.

   → If the gas properties stay constant throughout the calculation, it has to be specified only once.

26.2.4.6. Cylinder Charge

This dialog serves to define the global cylinder charge properties. These are necessary **for Spray Calculation** option in Nozzle elements. Within **Cylinder Charge Properties** dialog, the user may choose between three options:

1. **Global gas properties** [default].
   Cylinder Charge properties are taken from the **Global Gas Properties**.
2. **Cylinder Charge properties from Property Database**
   Cylinder charge properties are chosen from the menu in Property Database (refer to Chapter 26.2.4.8 for more information)
   - Gas name [name of specie]
3. **Variable cylinder charge properties.**
   The tabular values of cylinder charge properties as a function of pressure have to be specified for the following properties:
   - Reference density ρ [Denisty Units]
   - Kinematic viscosity ν [Kinematic Viscosity Units]

   Reference temperature and specific heat ratio must be defined additionally.

   General rules for the specification of cylinder charge properties:

→ Pressure values (first column) have to be specified in ascending order.

→ Intermediate values are bridged by linear interpolation. No extrapolation is performed.

→ If the pressure drops below the lowest value in the table (1^{st} row), the values of cylinder charge properties are taken from the 1^{st} row and kept constant until the pressure reaches the defined range again.

→ If the pressure exceeds the highest values in the table (last row), then the values of cylinder charge properties are taken from the last row and kept constant until the pressure drops down to the defined region again.

→ If the cylinder charge properties stay constant throughout the calculation, it has to be specified only once.

26.2.4.7. Solid Properties

This dialog serves to define the global solid properties. Solid properties can be **constant** or **variable** (temperature-dependent). Variable properties can be specified only through **Solid Property Database**. If the system contains no elastic elements, solid properties do not need to be defined.

Local Solid Properties are described in section 8.2.2.

1. **Solid properties from Property Database**

   Solid properties are chosen from the menu in Property Database (refer to Chapter 26.2.4.8 for more information)

   - Solid name [name of specie]
   - Temperature [Temperature Units]

2. **Constant solid properties.**

   - Densityρ [Density Units]
   - Young's modulus E [Stress Units]
   - Poisson's ratio ν [Ratio Units]
   - Thermal conductivity λ [Heat Conductivity Units]
   - Specific heat capacity c [Specific Heat Units]

26.2.4.8. Property Database

This dialog serves to add, extract, modify and otherwise manipulate the properties in the Property Database.

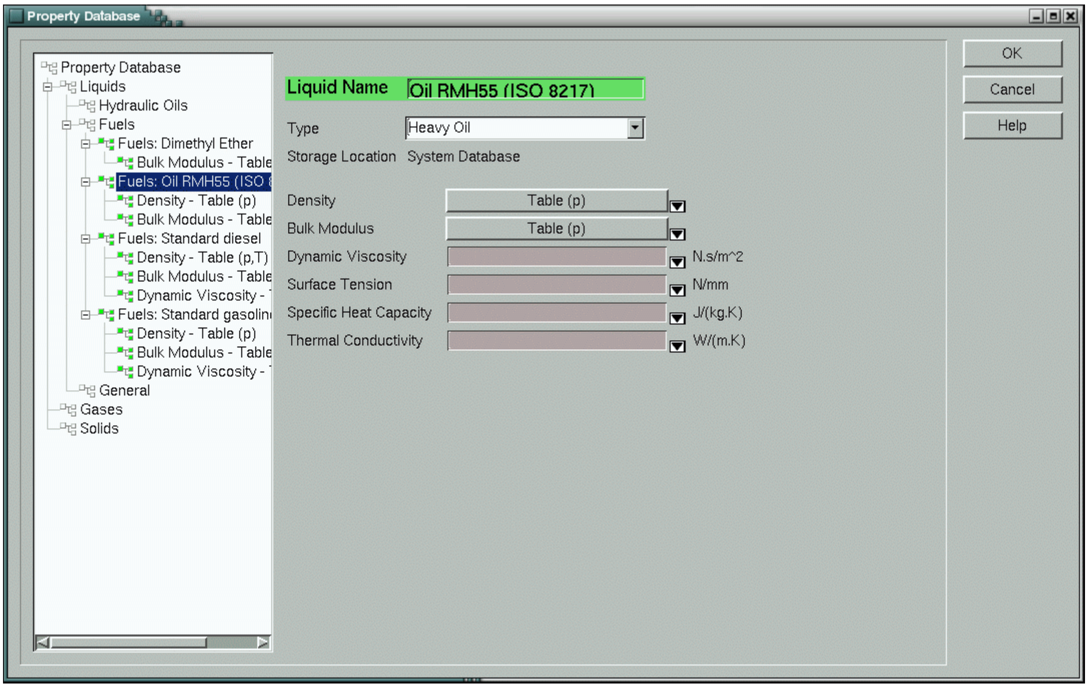


Figure 267: Property Database Dialog

Property Database	

Liquids
• **Hydraulic Oils**
  • **Hydraulic Oil Specie 1**
  • **Hydraulic Oil Specie 2**
  • **...**
• **Fuels**
  • **Fuel Specie 1**
  • **Fuel Specie 2**
  • **...**
• **General**
  • **General Liquid 1**
  • **General Liquid 2**
  • **...**
	Add new liquid in Property Database:
	1. In dialog tree select subgroup under which new liquid will be stored (Hydraulic Oils, Fuels or General)
	2. Right-click on dialog tree area
	3. Click New command
	Clone liquid:
	1. In dialog tree select the liquid you want to clone
	2. Right-click on dialog tree area
	3. Click Clone command
	Delete liquid from fluid database:
	1. In dialog tree select the liquid you want to delete
	2. Right-click on dialog tree area
	3. Click Delete command

Liquid Name	Specify unique species name under which liquid properties will be stored into property database.
Type	Specify type of the liquid.

<table>
<tr><td>Storage Location</td><td>Storage location may be:
Model
The data is stored in the model file.
User Database
The data is stored in the private storage space of the site in the file User.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_USERHOME).
Site Database
The data is stored in a shared location for all users of a site in the file Site.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_SITEHOME).
System Database
The data is stored in the installation path of the SW in the file System.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_HOME).
Default storage location is in the Model.

Move To liquid from one Storage Location to another one:
1. In dialog tree select the liquid you want to move
2. Right-click on dialog tree area
3. Click Move To command
4. In Move To dialog box define New Storage Location
5. Click OK</td></tr>
<tr><td>Density</td><td>Specify liquid density.</td></tr>
<tr><td>Bulk Modulus</td><td>Specify liquid bulk modulus.</td></tr>
<tr><td>Dynamic Viscosity</td><td>Specify liquid dynamic viscosity.</td></tr>
<tr><td>Surface Tension</td><td>Specify liquid surface tension.</td></tr>
<tr><td></td><td>Each property consists of selection field that allows various input types for the property:
constant
table (property depends on one value: pressure or temperature)
map (property depends on two values: pressure and temperature)
formula (function)</td></tr>
</table>

• **Gases**
  • **Gas 1**
  • **Gas 2**
  • **...**
	Add new gas in Property Database:
	1. Right-click on dialog tree area
	2. Click New command
	Clone gas:
	1. In dialog tree select the gas you want to clone
	2. Right-click on dialog tree area
	3. Click Clone command
	
	Delete gas from property database:
	1. In dialog tree select the gas you want to delete
	2. Right-click on dialog tree area
	3. Click Delete command

Gas Name	Specify unique species name under which gas properties will be stored into property database.
Type	Specify type of the gas.
Storage Location	**Storage location may be:** • Model The data is stored in the model file. • User Database The data is stored in the private storage space of the site in the file User.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_USERHOME). • Site Database The data is stored in a shared location for all users of a site in the file Site.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_SITEHOME). • System Database The data is stored in the installation path of the SW in the file System.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_HOME). **Default storage location is in the Model.** **Move To gas from one Storage Location to another one:** 1. In dialog tree select the gas you want to move 2. Right-click on dialog tree area 3. Click Move To command 4. In Move To dialog box define New Storage Location 5. Click OK
Density	Specify gas density.

Dynamic Viscosity	Specify gas dynamic viscosity.
	Each property consists of selection field that allows various input types for the property: • constant • table (property depends on one value: pressure or temperature) • map (property depends on two values: pressure and temperature)
Solids • **Solid 1** • **Solid 2** • **...**	**Add new solid in Property Database:** 1. Right-click on dialog tree area 2. Click New command **Clone solid:** 1. In dialog tree select the solid you want to clone 2. Right-click on dialog tree area 3. Click Clone command **Delete solid from property database:** 1. In dialog tree select the gas you want to delete 2. Right-click on dialog tree area 3. Click Delete command
Solid Name	Specify unique species name under which gas properties will be stored into property database.
Type	Specify type of the solid.

Storage Location	**Storage location may be:** • Model The data is stored in the model file. • User Database The data is stored in the private storage space of the site in the file User.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_USERHOME). • Site Database The data is stored in a shared location for all users of a site in the file Site.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_SITEHOME). • System Database The data is stored in the installation path of the SW in the file System.PDB. In the AWS environment, this refers to the part of the file system under $(AWS_HOME). **Default storage location is in the Model.** **Move To solid from one Storage Location to another one:** 1. In dialog tree select the solid you want to move 2. Right-click on dialog tree area 3. Click Move To command 4. In Move To dialog box define New Storage Location 5. Click OK
Density	Specify solid density.
Young's Modulus	Specify solid Young's modulus..
Poisson's Ratio	Specify solid Poisson's ratio.
Thermal Conductivity	Specify solid thermal conductivity.
Specific Heat	Specify solid specific heat.
	Each property consists of selection field that allows various input types for the property: • constant • table (property depends on one value: pressure or temperature) • map (property depends on two values: pressure and temperature)

26.2.5. Export

The Export pull-down menu contains the following item:

FIRE Flow/Spray	This activates BOOST Hydsim interfaces to 3D CFD code: **AVL FIRE Nozzle Flow and FIRE Spray**. Refer to Section 26.2.5.1

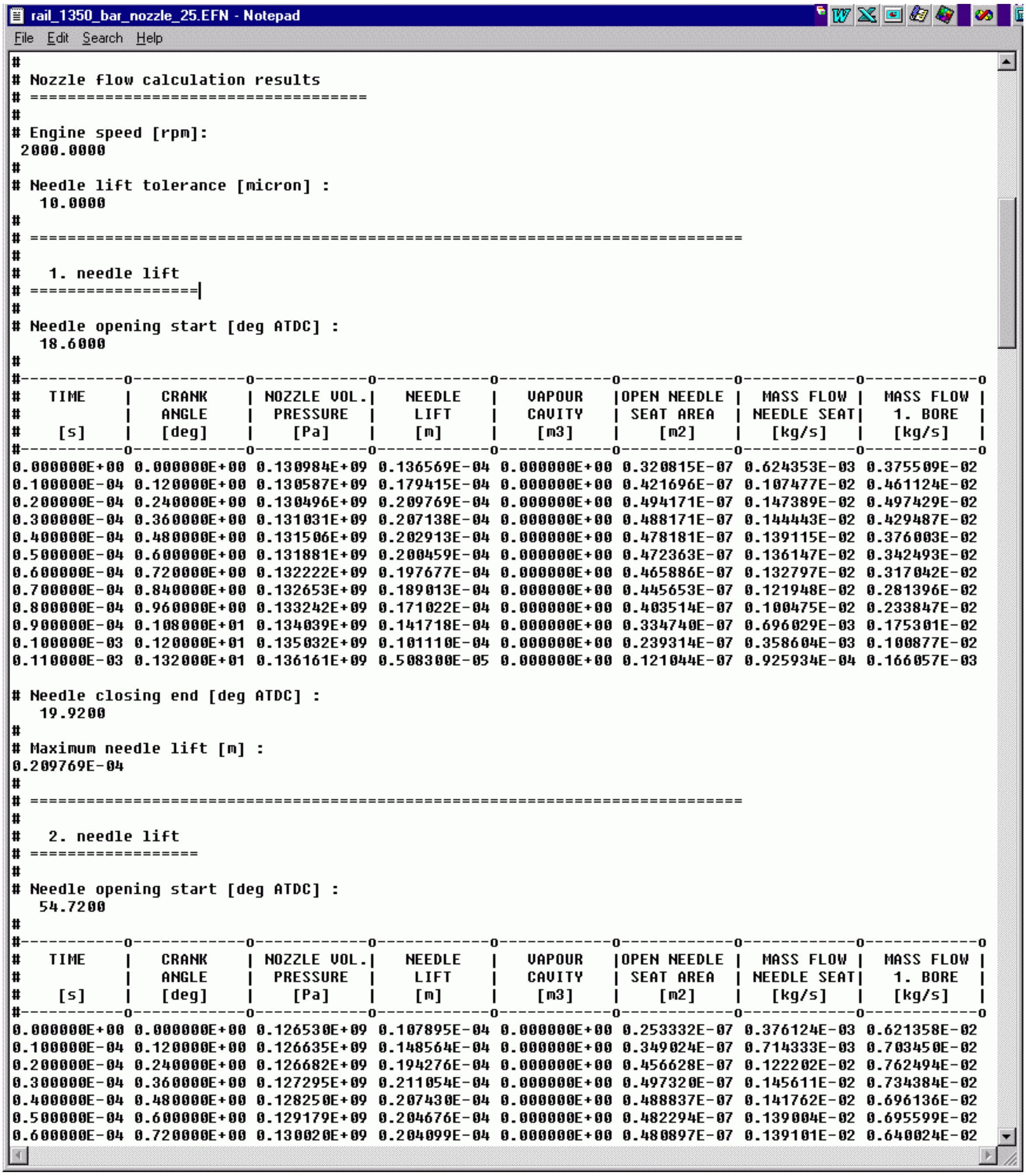
26.2.5.1. FIRE Nozzle Flow/Spray

Dialog Elements:

Needle lift tolerance open/closed)	Specify needle lift tolerance at which output of BOOST Hydsim results has to start/end.
Type of nozzle + ID number + "element name"	Click the corresponding box (boxes) to select the nozzle(s) for which interface files have to be produced.

For each selected nozzle element, two separate interface files (ASCII) are generated during a BOOST Hydsim calculation. The first file is the **Nozzle Flow** interface file (extension *.EFN). Part of this file is shown in Figure 268, which contains all data which could be necessary for the further 3D calculation at any position in the nozzle chamber: from the top (nozzle bores) till the bottom (needle seat).

Output data in the interface file is the following: nozzle type, geometric data, engine speed, needle opening start (deg BTDC), needle lift, nozzle chamber pressure, open area at needle seat, mass flow rate at input (nozzle bores) and output (needle seat). Output is provided as a function of time and crank angle. Furthermore, output is generated only for needle lift greater or equal than the tolerance specified in the **FIRE Nozzle Flow/Spray** dialog.

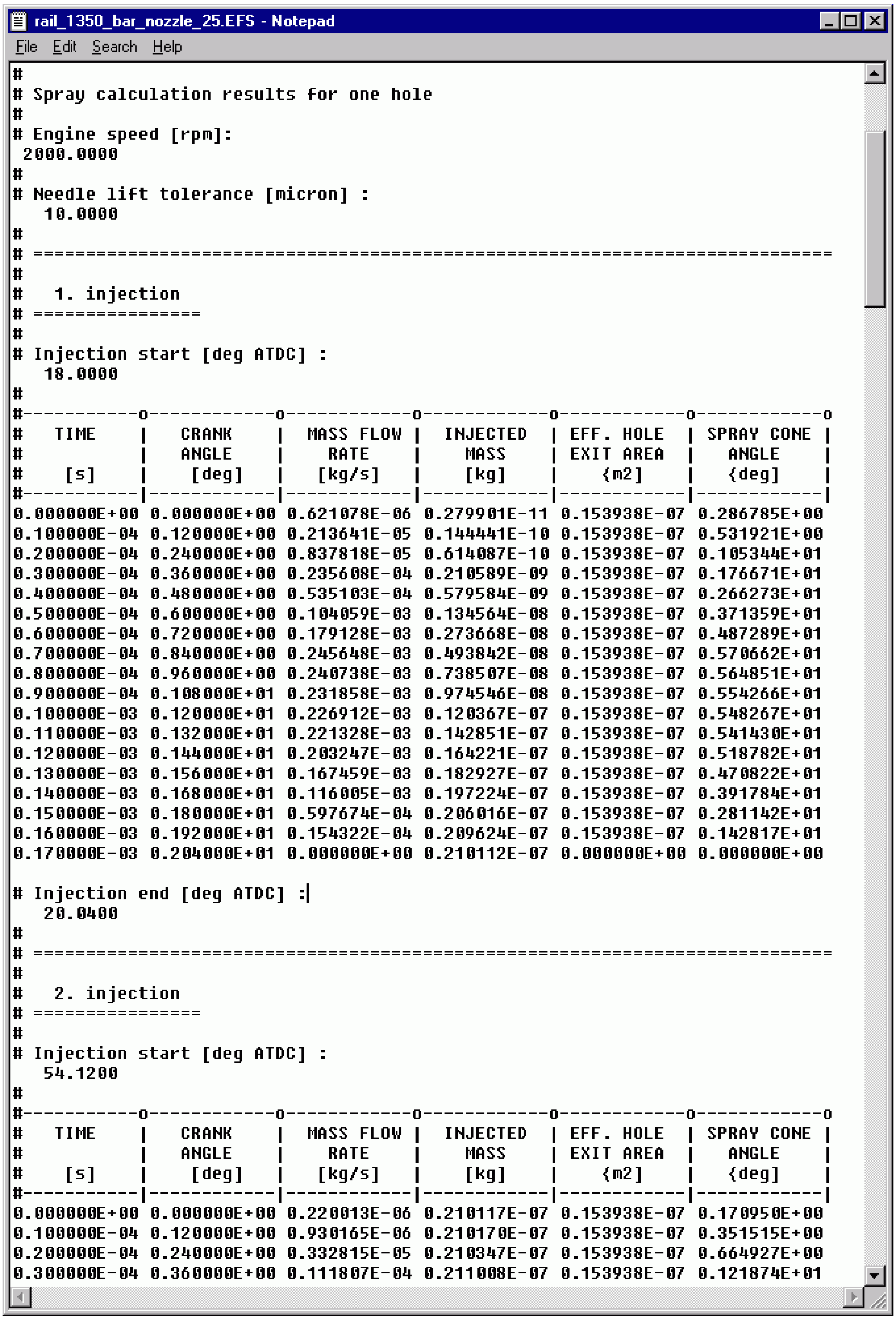


```
rail_1350_bar_nozzle_25.EFN - Notepad
File Edit Search Help
#
# Nozzle flow calculation results
# ====================================
#
# Engine speed [rpm]:
 2000.0000
#
# Needle lift tolerance [micron] :
   10.0000
#
# ============================================================================
#
#   1. needle lift
# ==================
#
# Needle opening start [deg ATDC] :
   18.6000
#
#-----------0------------0------------0------------0------------0------------0------------0------------0
#   TIME    |   CRANK    | NOZZLE VOL.|   NEEDLE   |   VAPOUR   |OPEN NEEDLE |  MASS FLOW |  MASS FLOW |
#           |   ANGLE    |  PRESSURE  |    LIFT    |   CAVITY   | SEAT AREA  | NEEDLE SEAT|   1. BORE  |
#    [s]    |   [deg]    |    [Pa]    |    [m]     |    [m3]    |    [m2]    |   [kg/s]   |   [kg/s]   |
#-----------0------------0------------0------------0------------0------------0------------0------------0
0.000000E+00 0.000000E+00 0.130984E+09 0.136569E-04 0.000000E+00 0.320815E-07 0.624353E-03 0.375509E-02
0.100000E-04 0.120000E+00 0.130587E+09 0.179415E-04 0.000000E+00 0.421696E-07 0.107477E-02 0.461124E-02
0.200000E-04 0.240000E+00 0.130496E+09 0.209769E-04 0.000000E+00 0.494171E-07 0.147389E-02 0.497429E-02
0.300000E-04 0.360000E+00 0.131031E+09 0.207138E-04 0.000000E+00 0.488171E-07 0.144443E-02 0.429487E-02
0.400000E-04 0.480000E+00 0.131506E+09 0.202913E-04 0.000000E+00 0.478181E-07 0.139115E-02 0.376003E-02
0.500000E-04 0.600000E+00 0.131881E+09 0.200459E-04 0.000000E+00 0.472363E-07 0.136147E-02 0.342493E-02
0.600000E-04 0.720000E+00 0.132222E+09 0.197677E-04 0.000000E+00 0.465886E-07 0.132797E-02 0.317042E-02
0.700000E-04 0.840000E+00 0.132653E+09 0.189013E-04 0.000000E+00 0.445653E-07 0.121948E-02 0.281396E-02
0.800000E-04 0.960000E+00 0.133242E+09 0.171022E-04 0.000000E+00 0.403514E-07 0.100475E-02 0.233847E-02
0.900000E-04 0.108000E+01 0.134039E+09 0.141718E-04 0.000000E+00 0.334740E-07 0.696029E-03 0.175301E-02
0.100000E-03 0.120000E+01 0.135032E+09 0.101110E-04 0.000000E+00 0.239314E-07 0.358604E-03 0.100877E-02
0.110000E-03 0.132000E+01 0.136161E+09 0.508300E-05 0.000000E+00 0.121044E-07 0.925934E-04 0.166057E-03

# Needle closing end [deg ATDC] :
   19.9200
#
# Maximum needle lift [m] :
0.209769E-04
#
# ============================================================================
#
#   2. needle lift
# ==================
#
# Needle opening start [deg ATDC] :
   54.7200
#
#-----------0------------0------------0------------0------------0------------0------------0------------0
#   TIME    |   CRANK    | NOZZLE VOL.|   NEEDLE   |   VAPOUR   |OPEN NEEDLE |  MASS FLOW |  MASS FLOW |
#           |   ANGLE    |  PRESSURE  |    LIFT    |   CAVITY   | SEAT AREA  | NEEDLE SEAT|   1. BORE  |
#    [s]    |   [deg]    |    [Pa]    |    [m]     |    [m3]    |    [m2]    |   [kg/s]   |   [kg/s]   |
#-----------0------------0------------0------------0------------0------------0------------0------------0
0.000000E+00 0.000000E+00 0.126530E+09 0.107895E-04 0.000000E+00 0.253332E-07 0.376124E-03 0.621358E-02
0.100000E-04 0.120000E+00 0.126635E+09 0.148564E-04 0.000000E+00 0.349024E-07 0.714333E-03 0.703450E-02
0.200000E-04 0.240000E+00 0.126682E+09 0.194276E-04 0.000000E+00 0.456628E-07 0.122202E-02 0.762494E-02
0.300000E-04 0.360000E+00 0.127295E+09 0.211054E-04 0.000000E+00 0.497320E-07 0.145611E-02 0.734384E-02
0.400000E-04 0.480000E+00 0.128250E+09 0.207430E-04 0.000000E+00 0.488837E-07 0.141762E-02 0.696136E-02
0.500000E-04 0.600000E+00 0.129179E+09 0.204676E-04 0.000000E+00 0.482294E-07 0.139004E-02 0.695599E-02
0.600000E-04 0.720000E+00 0.130020E+09 0.204099E-04 0.000000E+00 0.480897E-07 0.139101E-02 0.640024E-02
```

Figure 268: Interface File BOOST Hydsim – FIRE Nozzle Flow

The second file is the **Nozzle Spray** interface file (extension *.EFS). Part of this file is shown in Figure 269. There BOOST Hydsim writes out the calculation results at spray hole outlet. This implies that the nozzle flow is modeled with 1D BOOST Hydsim model. This output file is aimed to be used for further 3D spray calculation in injection chamber.


rail_1350_bar_nozzle_25.EFS - Notepad

File Edit Search Help

```
#
# Spray calculation results for one hole
#
# Engine speed [rpm]:
 2000.0000
#
# Needle lift tolerance [micron] :
   10.0000
#
# ============================================================================
#
#   1. injection
# ===============
#
# Injection start [deg ATDC] :
   18.0000
#
#-----------o------------o------------o------------o------------o------------o
#    TIME    |    CRANK   |  MASS FLOW |  INJECTED  | EFF. HOLE  | SPRAY CONE |
#            |    ANGLE   |    RATE    |    MASS    | EXIT AREA  |   ANGLE    |
#     [s]    |    [deg]   |   [kg/s]   |    [kg]    |   {m2]     |   {deg]    |
#-----------|------------|------------|------------|------------|------------|
0.000000E+00 0.000000E+00 0.621078E-06 0.279901E-11 0.153938E-07 0.286785E+00
0.100000E-04 0.120000E+00 0.213641E-05 0.144441E-10 0.153938E-07 0.531921E+00
0.200000E-04 0.240000E+00 0.837818E-05 0.614087E-10 0.153938E-07 0.105344E+01
0.300000E-04 0.360000E+00 0.235608E-04 0.210589E-09 0.153938E-07 0.176671E+01
0.400000E-04 0.480000E+00 0.535103E-04 0.579584E-09 0.153938E-07 0.266273E+01
0.500000E-04 0.600000E+00 0.104059E-03 0.134564E-08 0.153938E-07 0.371359E+01
0.600000E-04 0.720000E+00 0.179128E-03 0.273668E-08 0.153938E-07 0.487289E+01
0.700000E-04 0.840000E+00 0.245648E-03 0.493842E-08 0.153938E-07 0.570662E+01
0.800000E-04 0.960000E+00 0.240738E-03 0.738507E-08 0.153938E-07 0.564851E+01
0.900000E-04 0.108000E+01 0.231858E-03 0.974546E-08 0.153938E-07 0.554266E+01
0.100000E-03 0.120000E+01 0.226912E-03 0.120367E-07 0.153938E-07 0.548267E+01
0.110000E-03 0.132000E+01 0.221328E-03 0.142851E-07 0.153938E-07 0.541430E+01
0.120000E-03 0.144000E+01 0.203247E-03 0.164221E-07 0.153938E-07 0.518782E+01
0.130000E-03 0.156000E+01 0.167459E-03 0.182927E-07 0.153938E-07 0.470822E+01
0.140000E-03 0.168000E+01 0.116005E-03 0.197224E-07 0.153938E-07 0.391784E+01
0.150000E-03 0.180000E+01 0.597674E-04 0.206016E-07 0.153938E-07 0.281142E+01
0.160000E-03 0.192000E+01 0.154322E-04 0.209624E-07 0.153938E-07 0.142817E+01
0.170000E-03 0.204000E+01 0.000000E+00 0.210112E-07 0.000000E+00 0.000000E+00

# Injection end [deg ATDC] :
   20.0400
#
# ============================================================================
#
#   2. injection
# ===============
#
# Injection start [deg ATDC] :
   54.1200
#
#-----------o------------o------------o------------o------------o------------o
#    TIME    |    CRANK   |  MASS FLOW |  INJECTED  | EFF. HOLE  | SPRAY CONE |
#            |    ANGLE   |    RATE    |    MASS    | EXIT AREA  |   ANGLE    |
#     [s]    |    [deg]   |   [kg/s]   |    [kg]    |   {m2]     |   {deg]    |
#-----------|------------|------------|------------|------------|------------|
0.000000E+00 0.000000E+00 0.220013E-06 0.210117E-07 0.153938E-07 0.170950E+00
0.100000E-04 0.120000E+00 0.930165E-06 0.210170E-07 0.153938E-07 0.351515E+00
0.200000E-04 0.240000E+00 0.332815E-05 0.210347E-07 0.153938E-07 0.664927E+00
0.300000E-04 0.360000E+00 0.111807E-04 0.211008E-07 0.153938E-07 0.121874E+01
```

Figure 269: Interface File BOOST Hydsim – FIRE Spray

The output data in this interface file is the following: nozzle type, number of spray holes, hole diameter, engine speed, injection start (deg ATDC), mass flow rate and injected mass for one hole, effective hole exit area and spray cone angle. The output is generated only for needle lift greater or equal than the tolerance specified in the **FIRE Nozzle Flow/Spray** dialog. Results are provided as a function of time and crank angle.

26.2.6. Simulation

The Simulation PullDownMenu contains the following items:

Run:	Click to open the **Run** dialog (start calculation). Refer to Section 26.2.6.1.
Restart:	Click to continue simulation of the previously calculated model from the saved position. Refer to Section 26.2.6.2.
Control:	Click to open the **Calculation Control** dialog. Refer to Section **26.2.6.3**
Mode::	Click to open the **Mode** dialog. Refer to Section 26.2.6.4.
Boundary Data	Click to open the **Boundary Data** dialog. Refer to Section 26.2.6.5.
Search Adjust:	Definition of the list of target functions (sets). Each target implies an independent **Search/Adjust** calculation run. Refer to Section 26.2.6.6.
Animation	Click to open the **Animation/Nozzle Flow** dialog. Refer to Section 26.2.6.7.
Status:	Click to open the **Simulation Status** window
View Logfile:	Click to open the **View Logfile** window

26.2.6.1. Run

In the **Run** dialog the desired variants from the list of **Cases** can be selected. Click the **Run** button to start the simulation with all selected cases.

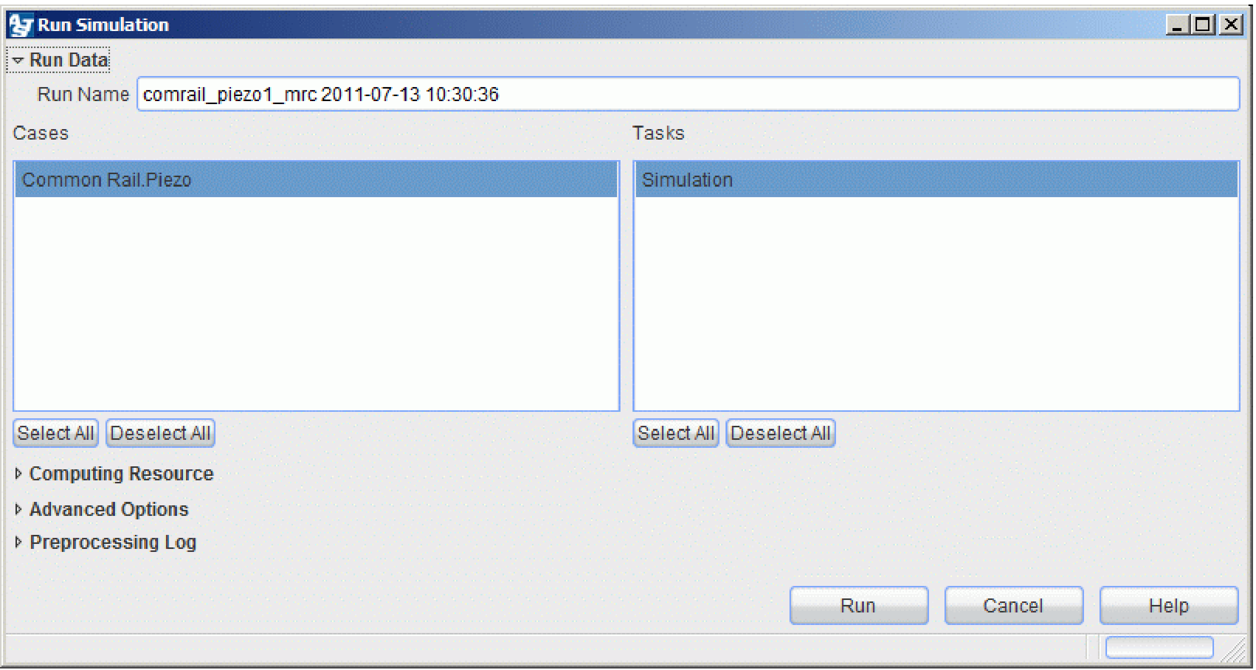


Figure 270: Run Dialog

26.2.6.2. Restart

The **Restart** option performs further simulation of the previously calculated model from the saved position. This position has been defined previously in the **Calculation Control** dialog through parameter **Save interval for restart** (refer to Section 26.2.6.3).

If **Restart** is clicked, BOOST Hydsim searches for the **Start-file** with the **model/case** name and extension `.STA` and executes it. If **Start-file** is not found, an error message will be produced.

The **Restart** option allows the following modifications in the original BOOST Hydsim model:

- Extension of **Time/Ref. Angle interval** in **Calculation Control** dialog (necessary condition). However, **Time/Ref. Angle step** cannot be changed.
- Modification of **Hydraulic, Mechanical**, and/or **Hydromechanical Boundaries** in the model using **Modify Parameter** function.
- Definition of other output parameters for all elements using **Store Results** function.
- Definition of the new **Save interval for restart** in **Calculation Control** dialog if e.g. further calculation with the new **Start-file** is expected to be performed.

26.2.6.3. Calculation Control

The **Calculation Control** dialog is shown in Figure 271.

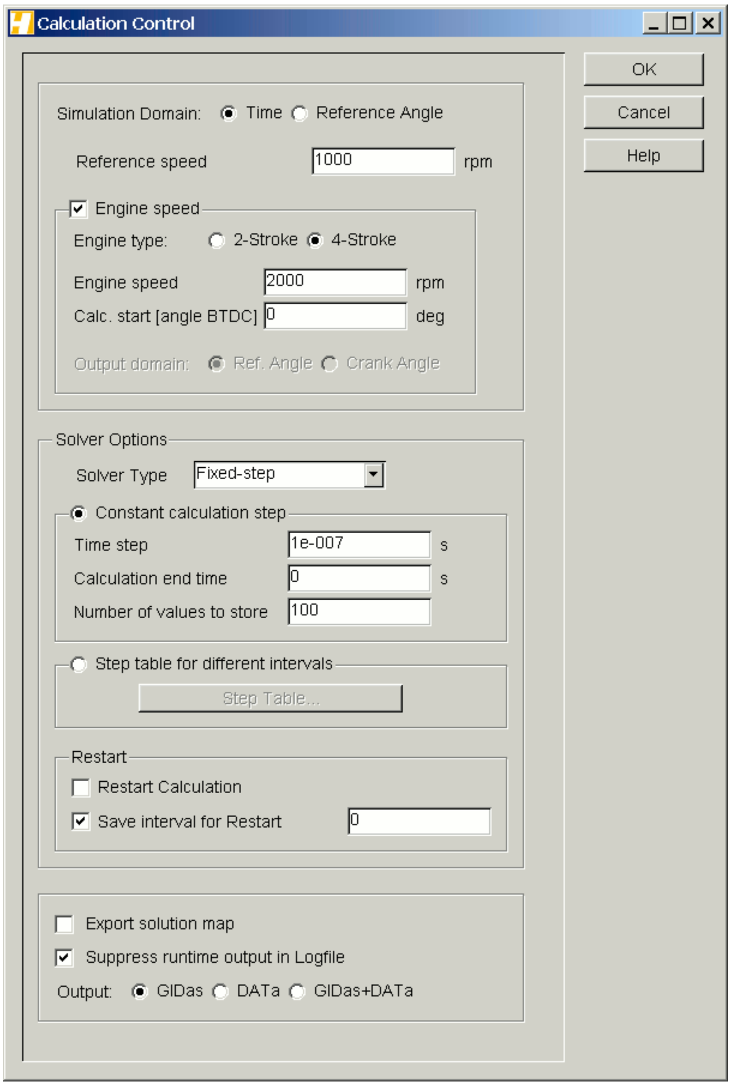


Figure 271: Calculation Control Dialog

Dialog Elements:

Simulation Domain:	Select **Time** or **Reference Angle**. If **Time** domain is selected, all output data will be stored as a function of time only.

Reference Speed	If **Reference Angle** domain is selected, all output results will be stored as a function of time and reference angle. **Reference Angle** domain requires positive **Reference speed** to be specified. BOOST Hydsim performs the transformation from **Time** to **Reference Angle** domain and vice versa according to the formula: $\varphi = 6 n_{ref} t,$ where: φ reference angle [deg] n_{ref} reference speed [rpm] t time [s]
Engine speed	If activated, positive **Engine speed** has to be specified.
Engine type	Select **2-Stroke** or **4-Stroke**.
Engine speed	**Engine speed** is required if some input data is available in crank angle domain or if recalculation from time to crank angle domain has to be performed.
Calculation start [angle BTDC]	Specify **calculation start** as angle before top dead center (BTDC) of firing.
Output domain	Select **Ref. Angle** or **Crank Angle** for the storage of calculation results. If **Reference Angle** simulation domain is selected, then: • For output domain **Ref. Angle** all results will be stored as a function of time and reference angle. • For output domain **Crank Angle** all results will be stored as a function of time and crank angle.

Solver Options	
Solver Type	**Fixed-step** Selects an constant time step solver (default), based on the explicit Runge-Kutta-Gill 4th order scheme.
Constant calculation step	**Time step / Reference angle step** Specify time step Δt (for active **Time** domain) or reference angle step $\Delta\varphi$ (for active **Reference angle** domain). **Calculation end time / End reference angle** Specify time interval t_{end} (for active **Time** domain) or reference angle interval φ_{end} (for active **Reference angle** domain). **Number of values to store** Specify the number of values to be stored on output. It cannot exceed the number of calculation steps N obtained from: $N = \frac{t_{end}}{\Delta t}$ for **Time** domain $N = \frac{\varphi_{end}}{\Delta\varphi}$ for **Reference angle** domain The number of values to be stored should be estimated according to the required accuracy of output curves. Normally it should not exceed 1000.
Step table for different intervals	**Step Table** Press this bar to specify step table (refer to Section 26.2.6.3.1).
Restart	**Restart Calculation** If checked, this implies continuation (restart) of previously performed and saved calculation. The new calculation is restarted with initial conditions taken from a **Start-file**. If not checked (**Restart** inactive), the standard calculation will be performed (ith initial values specified at initial conditions). **Note:** Newly specified calculation end time/angle must be greater than restart time/angle. **Save interval for restart** Select to save model data on a special **Start-file** with the extension `.STA`. If activated, enter the position within **Number of values to store** at which the data have to be saved. **Start-file** can be further used for the calculation **Restart** from the saved position. For each multiple step of the **Save interval for restart**, the **Start-file** will be created again, i.e. it will overwrite the previous Start-file.
Export solution map	Select to export a solution map file with extension `.SMP`. This file contains information on integration history in a specific format. It should be used only by an experienced user for troubleshooting (if necessary).

Solver Type	**Variable-step** Selects a variable step solver, based on implicit Backward Differentiation Formula (BDF) method.

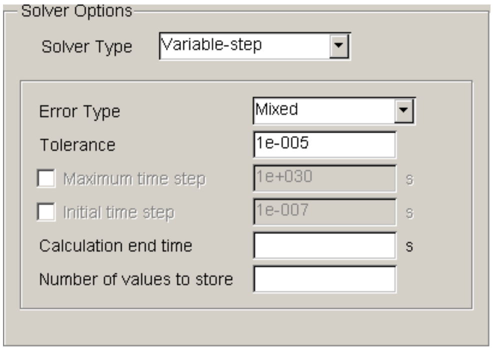


Figure 272: Variable Step Solver

Error Type	Select which type of tolerance will be used for specifying solution accuracy. **Mixed** A mixed tolerance will be used to control the accuracy of solution, based on the formula: $$\lvert e \rvert < r_{tol} \cdot \lvert y_i \rvert + a_{tol} \ ;$$ where e is the local error, r_{tol} and a_{tol} are the relative and absolute tolerances, and y_i is the state variable value. This is the recommended tolerance type. **Absolute** Only the absolute tolerance will be used to control the accuracy of solution, according to the formula: $$\lvert e \rvert < a_{tol}$$ **Relative** Only the relative tolerance will be used to control the accuracy of the solution, according to the formula: $$\lvert e \rvert < r_{tol} \cdot \lvert y_i \rvert$$ This is the most demanding type of tolerance (should be used only in special cases).
Tolerance	Specify tolerance size. For stiff systems, the recommended value is 1.d-5, otherwise the value 1.d-4 should be used
Maximum time step (optional)	Input the upper limit for time step size; this prevents choosing of a too high time step which can lead to loss of accuracy in certain periods.
Initial time step (optional)	Specify the initial time step size; if not defined, the solver will calculate it internally.
Calculation end time / End reference angle	Specify time interval t_{end} (for active **Time** domain) or reference angle interval φ_{end} (for active **Reference angle** domain).

Number of values to store	Specify number of values to be stored on output. It cannot exceed the number of calculation steps N obtained from: $N = \frac{t_{end}}{\Delta t}$ for **Time** domain; $N = \frac{\varphi_{end}}{\Delta\varphi}$ for **Reference angle** domain. The number of values to be stored should be estimated according to the required accuracy of output curves. Normally it should not exceed 1000.
Suppress runtime output in Logfile	Check to suppress the runtime table (generated by HYDSIM kernel) in the **View Logfile** window.
Output	Select type of the output files. **GIDas** file (with extension .GID) is absolutely necessary for the post-processing of the calculation results with the GUI post-processor IMPRESS Chart. **DATa** file (with extension `.dat`) is useful for viewing the results with external graphical programs. Both GIDas and DATa files are of ASCII type.

26.2.6.3.1. Variable Calculation Step

This dialog serves to define simulation intervals with different calculation steps.

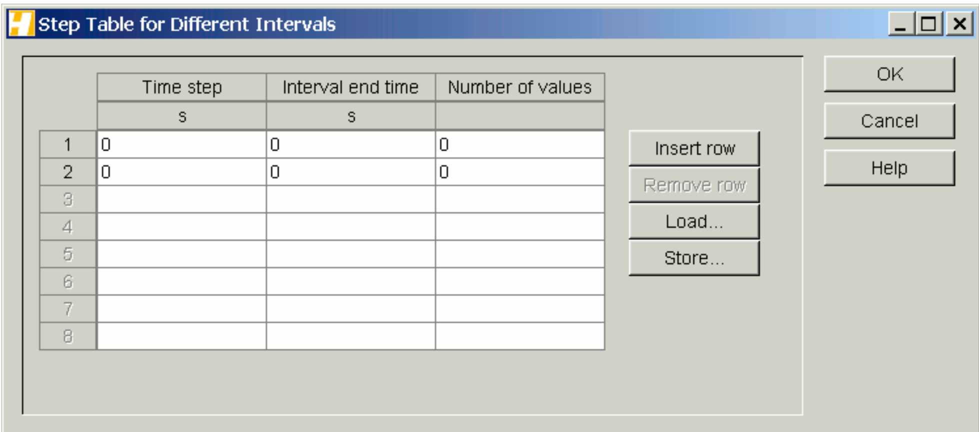


Figure 273: Variable Calculation Step Dialog

Description of input data:

Table	Specify time step Δt_i (for active **Time** domain) or reference angle step $\Delta\varphi_i$ (for active **Reference angle** domain) in the first column. Specify interval end time t^i_{end} (for active **Time** domain) or end reference angle φ^i_{end} (for active **Reference angle** domain) in the second column. Specify number of values to be stored on output for each calculation interval in the third column. It cannot exceed the number of calculation steps N obtained from:

	For **Time** domain: $N^i = \frac{t^i_{end} - t^{i-1}_{start}}{\Delta t_i}$; $t^i_{start} = t^{i-1}_{end}$; $t^1_{start} = 0.0$ For **Reference angle** domain: $N^i = \frac{\varphi^i_{end} - \varphi^{i-1}_{start}}{\Delta \varphi_i}$; $\varphi^i_{start} = \varphi^{i-1}_{end}$; $\varphi^1_{start} = 0.0$ Number of values to be stored should be estimated according to the required accuracy of output curves.

26.2.6.4. Mode

Description:

CALCULATION TASK		
Isothermal flow (weakly compressible)	Temperature is not a system variable: it is kept constant through the whole calculation. For each hydraulic element different temperature can be defined. It may be:	
	* **global constant temperature**	- If Fluid Properties from Property Database are active in Global Fluid Properties dialog box, it is treated as **Initial Temperature**.
		- If Constant or Variable Fluid Properties are active in Global Fluid Properties dialog box, it is treated as **Reference Temperature**.
	* **local constant temperature**	- If Local Fluid Properties from Property Database are active in Local Fluid Properties dialog box, it is treated as **Initial Temperature**.
	Fluid properties are the function of the user-defined (constant) temperature in each hydraulic element.	
Thermal flow (weakly compressible)	For Thermal flow option temperature is a system variable. Pressure and temperature in **Volume** group elements are calculated from the continuity equation, equation of state for compressible fluid and additional equation based on energy conservation law (refer to Governing Equations for Thermal Flow in Standard Volume). Temperature is calculated internally only for **Volume** and **Orifice** elements. External (boundary) temperature can be defined as a function of time/ref. angle in **Pressure/Temperature Boundary**. Other hydraulic elements (e.g. lines, throttles) simply transfer temperature from one end to the other (input->output or output->input) or import it from the neighboring **Volume**, **Orifice** or **Boundary** element according to the flow direction. Fluid properties are the function of internally-calculated temperature in each hydraulic element.	

TREATMENT OF INCONSISTENT CONNECTIONS	
In previous BOOST Hydsim versions hydraulic elements (line and orifice type) could be connected with volume or boundary elements only. The reason for this was that Line or Orifice element calculated flow rate on its ends from the pressures calculated from connected elements. In the case of two hydraulic elements connected directly (without a volume element in between) the program inserted a virtual volume between the two connected elements.	
Invert CONNECTIONS	The invert connections option is implemented for the rigorous handling of directly connected hydraulic elements with identical state variables. The program internally changes (inverts) the relevant state variable on one element end (e.g. from flow rate to pressure). This makes the connection consistent without any virtual volume between the lines. On the first line end the flow rate is calculated internally while the pressure is taken from the connected end of the second line. On the second line end the pressure is calculated internally while the flow rate is taken from the first line end (refer to Chapter Lines). This option is available only for direct connections between lines. Diameters of connected lines must be different and cross-sectional area of expansion/contraction at connections of one line must be equal to the corresponding cross-sectional area of the connected line. If it is not the case, the program will perform this automatically and issue a warning message(s).
Insert VIRTUAL VOLUMES	At calculation start a Virtual volume is introduced between the directly connected hydraulic elements (line-line, line-orifice etc.). Virtual volume size is defined according to the thumb rule: it is kept as small as possible to preserve the dynamic properties of the system. The initial pressure in Virtual volume is calculated as an average of the initial pressures of neighboring elements.
FLOW RATE CALCULATION FOR ORIFICE-TYPE ELEMENTS	
Fixed-step solver only	
Use original Runge-Kutta-Gill scheme	Flow rate in all hydraulic elements is calculated by a standard Runge-Kutta-Gill (RKG) integration procedure (in loop 2). **Note**: pressure in standard RKG scheme is calculated in loop 3.
Recalculate after Runge-Kutta-Gill loop 4 (for output)	Flow rate in 3 hydraulic elements (**Extended Sac Nozzle**, **Standard** and **Cavitating Orifice**) is recalculated after Runge-Kutta-Gill loop 4. **Note**: this recalculation is not a deviation from the Runke-Kutta-Gill scheme. It means that the flow rate is calculated one more time (after RKG loop 4) using last pressure values from loop 3. This is done only for output. In this way it is ensured that the flow rate sign corresponds to the pressure difference sign, even if the latter is very small (within numerical error range). However, flow magnitude error is proportional to the time step, i.e. it vanishes only with infinitesimally small step. For all remaining hydraulic elements, the flow rate is calculated by the standard Runge-Kutta-Gill integration procedure as before.

26.2.6.5. Boundary Data

Figure 274 shows the **Cycling of Boundary Data** dialog. To activate cycling of boundary data for each group of elements, select the appropriate button. If the global state is changed, this will not change the state in elements dialogs. Visually in each dialog the state will stay unchanged. The global button only has an affect on the local check boxes in the element dialogs at Simulation/Run, Run Sets or Restart command.

Activate the button to specify the values in the element table as a function of time/reference angle/crank angle only for one cycle. If more than one cycle is considered in the calculation, the program will copy data from table for each new cycle.

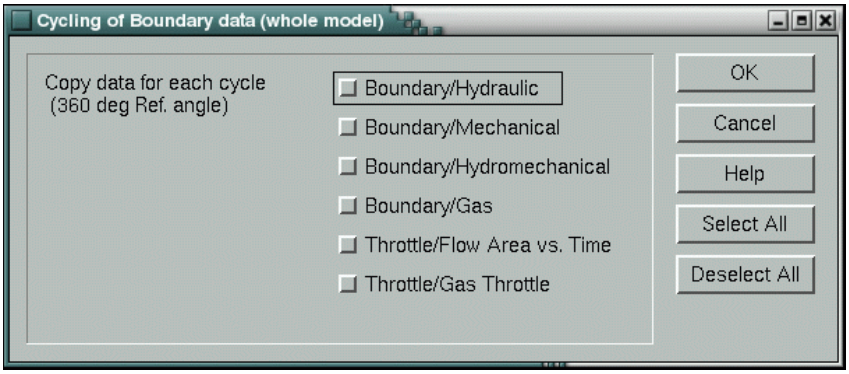


Figure 274: Cycling of Boundary Data Dialog

Boundary/Hydraulic correlates with two elements: Boundary/Pressure/Temperature and Boundary/Flow.

26.2.6.6. Search Adjust

This dialog serves to show a list of target functions (sets). Each target implies an independent **Search/Adjust** calculation run (1D Optimization).

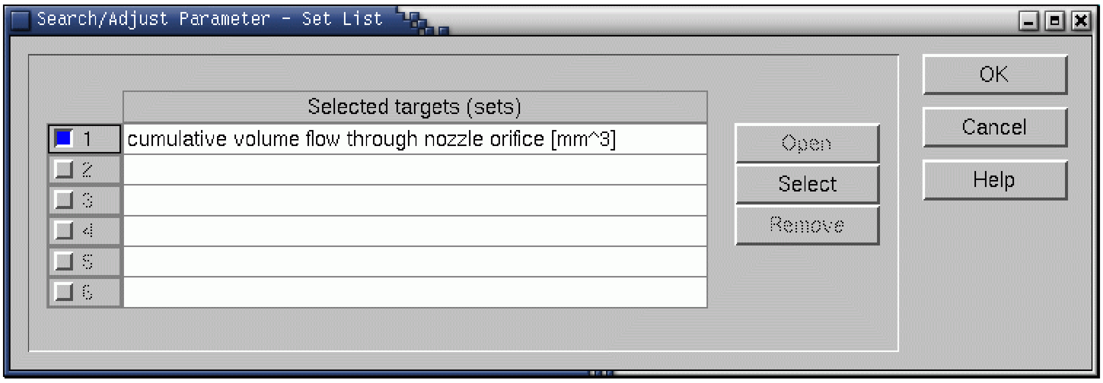


Figure 275: Search/Adjust Parameter – Set List

Dialog Elements:

Selected targets/sets	Displays a target list selected by the user.	
	Open	To open the selected set, highlight it and click **OK**. A **Dialog Search/Adjust - Definition of Selected Set** will pop up.
	Select	Press to add a new target from the **Select Target Dialog**.
	Remove	Press to remove the highlighted target from the target list.

Note: **Search/Adjust Parameter Set** associated with a specific target will be executed within BOOST Hydsim calculation only if the option box on the left side of the target (set) name is checked.

Note: Number of defined **Search/Adjust Parameter Sets** is unlimited. However, only up to six sets can be executed within one BOOST Hydsim run. They will be executed one after another, in the sequence as defined in the Set List dialog. Each new **Search/Adjust Set** will use the optimization results from previous set as initial values.

26.2.6.6.1. Selection of Targets (Objective Functions)

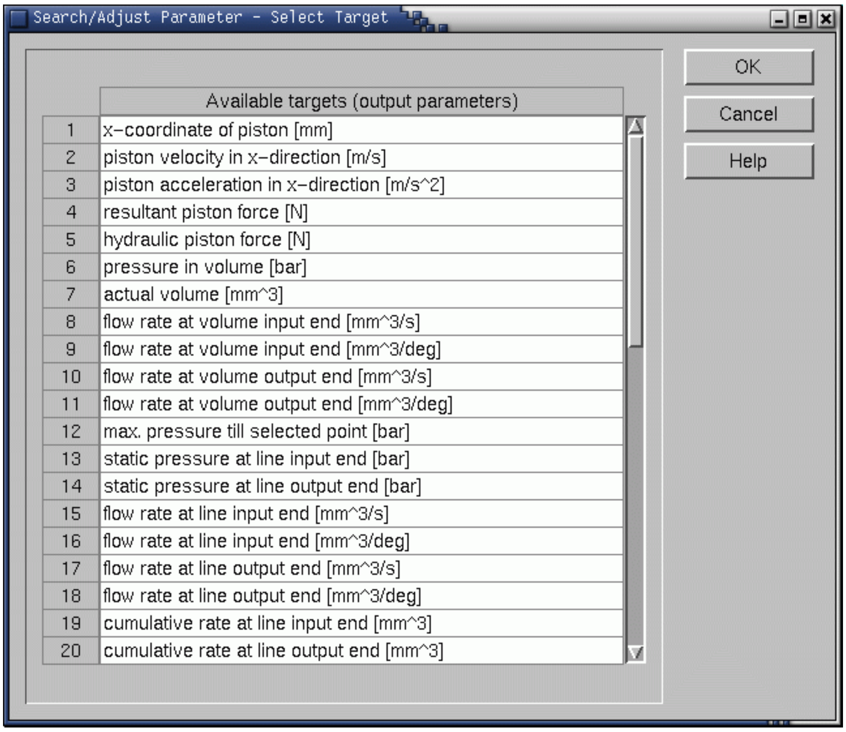


Figure 276: Search/Adjust Parameter – Select Target

Dialog Elements:

Available targets (output parameters)	To open a desired target, highlight it and click OK. The target will be immediately included into the **Selected Targets** list in **Search/Adjust Parameter – Set List** dialog. Each target can be added to the list of **Selected Targets** only once.

Target function is defined in Definition of Selected Set Dialogs (refer to Section 26.2.6.6.2).

Inside Search/Adjust algorithm, a one-dimensional regular falsi (bisection) method is used.

Search/Adjust procedure requires special care in choosing target function Q and adjusted parameters P1,...,Pn. There are many conditions (e.g. continuity, differentiability, etc.) imposed on the optimization function Q = f(P1,...,Pn). They has to be satisfied to achieve convergence, otherwise the process will diverge.

Furthermore, the search procedure is computationally very much involved, therefore it is not recommended to perform many Adjust Parameter runs within one BOOST Hydsim calculation.

Currently the following limits are imposed:

- Number of Adjust Parameter Sets within one calculation (6)
- Number of optimized parameter (20)
- Number of terms in target function (20)
- Number of iterations (20)

26.2.6.6.2. Definition of Selected Set

This dialog serves to define a target function, elements and their parameters to be adjusted (optimized) and additional data for **Search/Adjust** algorithm.

Highlight the selected target described in section 26.2.4.5 and press **Open**. The following window will appear:

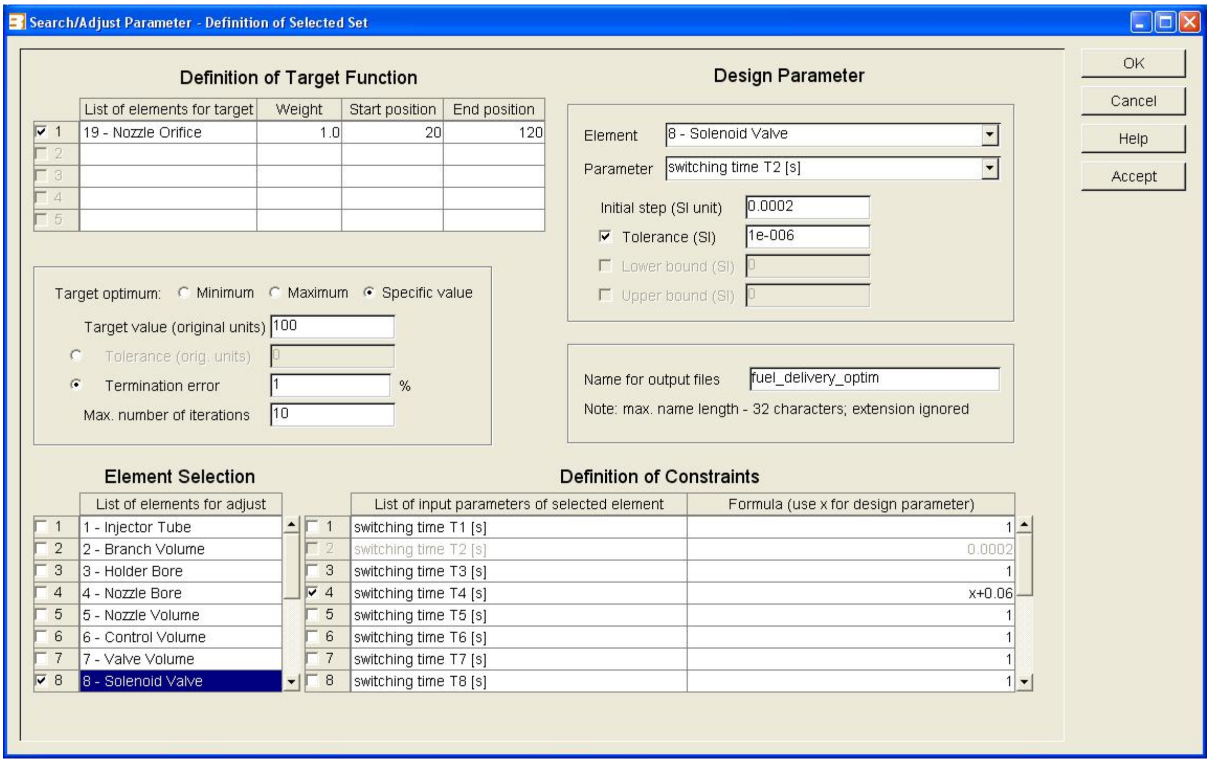


Figure 277: Search Adjust Parameters – Definition of Selected Set

Dialog Elements:

DEFINITION OF TARGET FUNCTION	Selection of elements for target function.	
	List of elements for target	Click a rectangular button on the left to include the selected element into the target function.
	Weight	Specify weighting factor for the element (refer to definition of target function below).

	Start position	Specify position in results array from which the output parameter value has to be evaluated.
	End position	Specify position in results array till which the output parameter value has to be evaluated.
Target function: Minimum		Not implemented yet
Target function: Maximum		Not implemented yet
Target function: Value		
	Target value (original units)	Specify value of target function (in original units).
	Tolerance (original units)	Specify absolute tolerance of the target function for the termination of the search algorithm.
	Termination error [%]	Specify relative error for the termination of the search algorithm.
	Max. number of iterations	Specify maximum number of iterations for search algorithm.
DESIGN PARAMETER	Selection of independent design variable (optimized parameter).	
	Element	Select an element with the design parameter
	Parameter	Select the design parameter from the list
	Initial step (SI unit)	Specify initial step for the design parameter (in SI units).
	Tolerance (SI)	Specify tolerance (minimal step) of design parameter in SI units (for discrete optimization)
	Lower bound (SI)	Not implemented yet
	Upper bound (SI)	Not implemented yet
Name for Output Files	Specify file name (without extension) for result files of the actual **Search Adjust Parameter Set**.	
ELEMENT SELECTION	Selection of elements with constrained (dependent) parameters.	
	List of elements for adjust	Click a rectangular button on the left side to include the element into the **Search/Adjust** list. Highlight the element to get a list of its adjustable parameters.
DEFINITION OF CONSTRAINTS	Selection of input parameters dependent on the design variable	
	List of input parameters of selected element	Click a rectangular button on the left side of parameter name to select the dependent parameter
	Formula (use *x* for design parameter)	Specify the formula (constraint) for the selected parameter. Design parameter notation in the formula is by convention *x* or *X*.

Target function Q_j is defined as follows:

$$Q_j = \sum_{i=1}^{n} \alpha_i A_{ij} \ , \tag{490}$$

where: α_i ; i=1,...,n weighting factor (*default* 1)

n number of elements in **List of elements for target**

A_{ij}; i=1,….,n Output parameter (defined in **Select Target** dialog) of highlighted element from **List of elements for target** at **Store position** j in result file.

If **Start position** is not specified or left 0, it is set to 1 (i.e. the first output parameter value is used.

If **End position** is not specified or left 0, the last output parameter value is used (at the last output step which may not necessarily coincide with the last calculation step).

Target value has to be specified in original units which are given in selected targets list in **Search/Adjust Parameter - Set List** dialog.

List of elements for target contains those elements of the actual model which contain selected target as output parameter.

List of elements for adjust contains all elements of the actual model. Some of them (e.g. **Boundary Conditions**) do not have adjustable parameters (by highlighting them the list of parameters will remain empty).

As **Search/Adjust** algorithm is one-dimensional, it may have only one design variable (independent parameter). Other selected parameters are dependent on the design variable by the user-defined analytical formula. This implies the solution of nonlinear algebraic equation

$$f(P_D) - Q = 0 \ , \tag{491}$$

where P_D is the design parameter and Q is the target function value.

The results of the last iteration (no matter whether converged or not) are stored on GIDas and/or DATa file (depending on what output is chosen in **Calculation Control**) using the name specified in **Name for output file**. Additionally, an iteration history file of GIDas type is produced (with the same name but another extension `.GAD`).

26.2.6.7. Animation/Nozzle Flow

The **Animation/Nozzle Flow** dialog is shown in Figure 278.

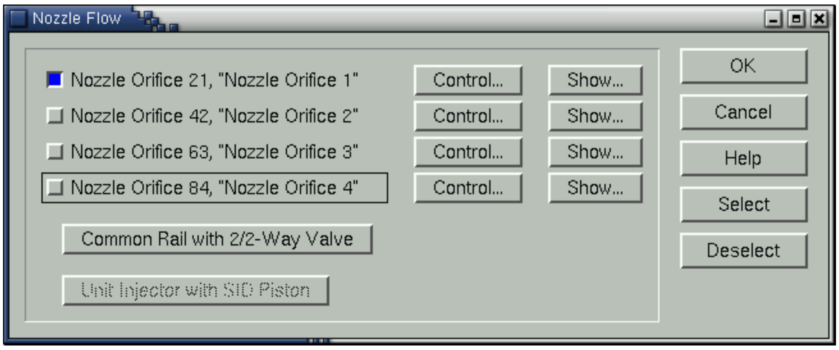


Figure 278: Nozzle Flow Animation Dialog

Dialog Elements:

Type of nozzle + ID number + "element name"	Click the corresponding box to select the nozzle(s) for which animation output file has to be produced (for PP3).
Control	Click to open the **Animation Control** dialog. Refer to Section 26.2.6.7.1
Show	Click to open **Show Nozzle Flow Animation** dialog and start animation with postprocessor PP3. **Note:** Before **Show Animation** can be started, a normal calculation **Run** with the activated animation has to be performed.
Common Rail with 2/2-Way Valve	Click to open **Common Rail with 2/2-Way Valve** dialog and define characteristic elements in common rail with 2/2-way valve system for each active nozzle. Refer to Section 26.2.6.7.2. **Note:** Obviously, it is impossible to set up animation of common-rail and unit injector within the same BOOST Hydsim model.
Unit Injector with SID Piston	Click to open **Unit Injector with SID Piston** dialog and define characteristic elements in unit injector with SID piston system for each active nozzle. Refer to Section 26.2.6.7.3.

26.2.6.7.1. Animation Control

Dialog Elements:

User-defined Animation Range	**Note:** If this button is not active, animation range will be equal to the calculation range.
Start / End table	Specified time/reference angle for the animation start/end.

26.2.6.7.2. Common Rail with 2/2-Way Valve

The animation dialog of **Common Rail with 2/2-Way Valve** is shown in Figure 279.

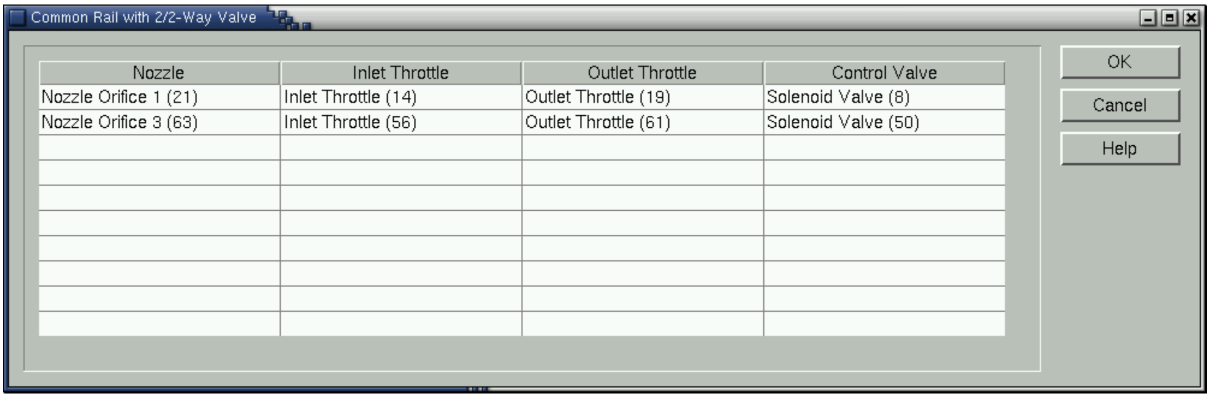


Figure 279: Animation Dialog of Common Rail with 2/2-Way Valve

Dialog Elements:

<table>
<tr><td>Nozzle</td><td>List of selected nozzles in Nozzle Flow dialog box.</td></tr>
<tr><td>Inlet Throttle</td><td>Select inlet throttle element of corresponding nozzle (for specific common rail 2/2-way valve model).</td></tr>
<tr><td>Outlet Throttle</td><td>Select outlet throttle element of corresponding nozzle (for specific common rail 2/2-way valve model).</td></tr>
<tr><td>Control Valve</td><td>Select control valve element of corresponding nozzle (for specific common rail 2/2-way valve model).</td></tr>
<tr><td>

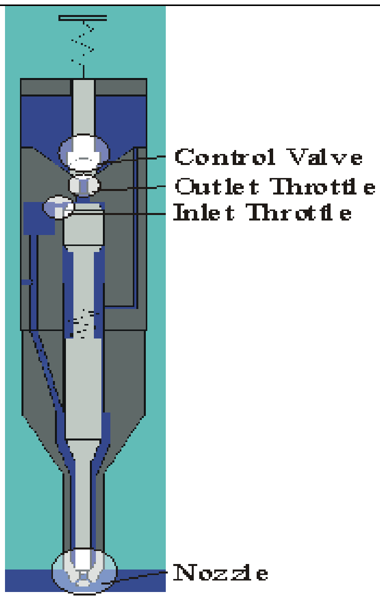


Figure 280: Common Rail with 2/2-Way Valve</td><td>

<u>Remarks:</u>

For animation of common rail with 2/2-way valve system, inlet throttle has to be selected.

- For animation of common rail with 2/2-way valve system with control valve, output throttle has to be selected.
- For animation of common rail with 2/2-way valve system including magnetic force, control valve has to be selected.
- Same throttle element can be selected only once.
- Same control valve element can be selected only once.

Refer to Figure 280.</td></tr>
</table>

26.2.6.7.3. Unit Injector with SID Piston

The animation dialog of **Unit Injector with SID Piston** dialog is shown in Figure 281.

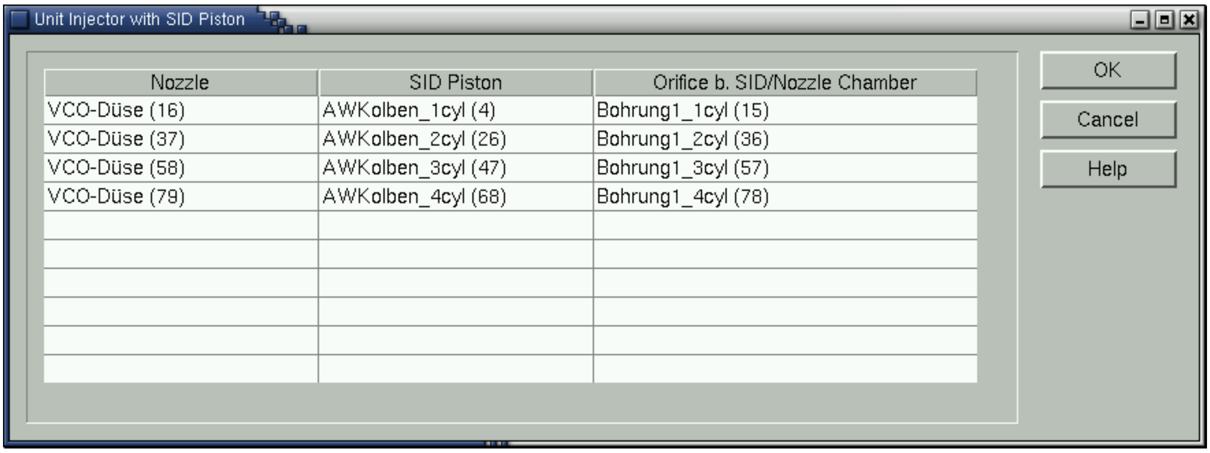
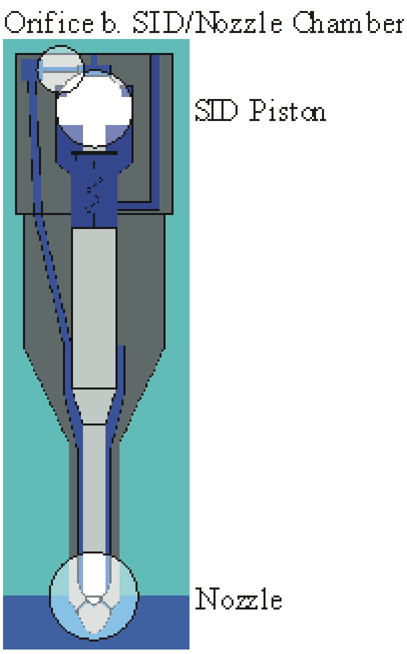


Figure 281: Animation Dialog of Unit Injector with SID Piston

Dialog Elements:

Nozzle	List of selected nozzles in Nozzle Flow dialog box.
SID Piston	Select SID Piston element of corresponding nozzle (for specific Unit Injector with SID Piston model).
Orifice b. SID/Nozzle Chamber	Select orifice or line element between volume on output side of SID piston and nozzle chamber (for specific Unit Injector with SID Piston model).
 **Figure 282: Unit Injector with SID Piston**	**Remarks:** For animation of Unit Injector with SID Piston, SID Piston element has to be selected. • Same SID Piston element can be selected only once. • Same orifice or line element can be selected only once. Refer to Figure 282

26.2.6.8. ACCI Interface/FIRE Nozzle

In the **ACCI Interface/FIRE Nozzle** dialog (Figure 283)the name and port number of the ACCI server used for the **BOOST Hydsim - FIRE** co-simulation have to be specified.

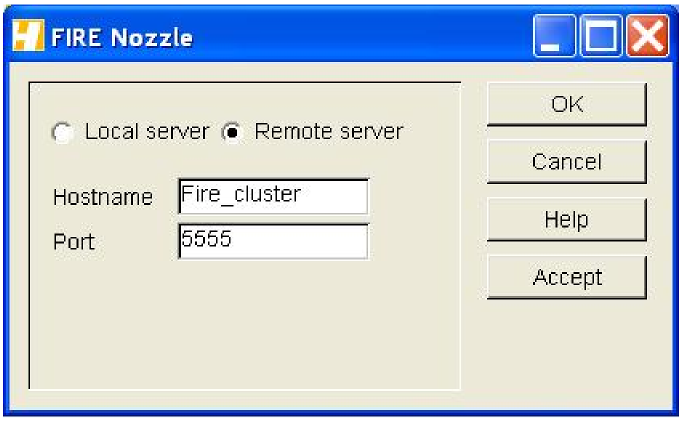


Figure 283: ACCI Interface/FIRE Nozzle Dialog

Dialog Elements:

Local server / Remote server	Click the corresponding button to select the Local computer or Remote server.
Hostname	Specify host-name of ACCI server (for **Remote Server** only)
Port	Specify port number for ACCI server (default 5555). **Note:** same port number has to be specified in the FIRE Workflow Manager.

26.2.6.9. ACCI Interface/BOOST Link

In the **ACCI Interface/BOOST Link** dialog (Figure 283) the name and port number of the ACCI server and exchange time/angle step have to be specified. Note that exchange step is a simulation period (cranktrain) variable related to the engine speed. Exchange step is unique for all BOOST Link elements in BOOST Hydsim model.

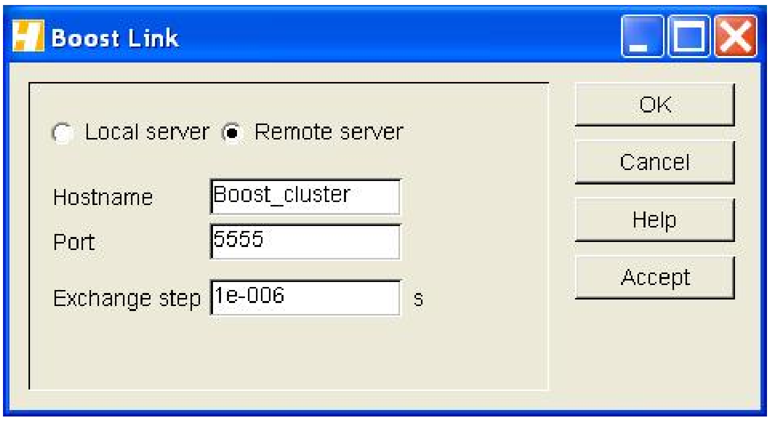


Figure 284: ACCI Interface/BOOST Link Dialog

Dialog Elements:

Local server / Remote server	Click the corresponding button to select the Local computer or Remote server.

Hostname	Specify host-name of ACCI server (for **Remote Server** only)
Port	Specify port number for ACCI server (default 5555). **Note:** same port number has to be specified in BOOST.
Exchange step	Specify time/angle step for data exchange (default 1e-6 s).

26.2.6.10. Units

AVL Workspace contains an integrated unit system. Among other features, this system transfers the input values from the input unit into the simulation unit.

Input unit has to be specified for each input parameter. This can be done by setting default input units in the **Units** main dialog accessible from the menu **Options**.

In this dialog the default units can be set for the user, site and system configuration. The settings can be copied from the upper configuration level e.g. from system settings.

All unit groups are listed in a tree. To change the default unit of a certain group, expand the tree for the required unit type. All available units for this unit group will be listed. The default unit is marked by gray rectangle in front of its name and followed by the test `Default` in brackets. To change the unit, double click on it. This will change the units for all input parameters of this unit group.

To change the unit of a single input parameter within a dialog, put the mouse pointer on the unit field and press the **right mouse button**. A unit selection box will pop up where all available units will be listed. Release the right mouse button on the desired unit. Recalculation of the parameter value will be performed automatically. If the same procedure is performed with the **left mouse button**, then the unit will be changed but recalculation of the parameter value will not be performed (parameter value will stay unchanged).

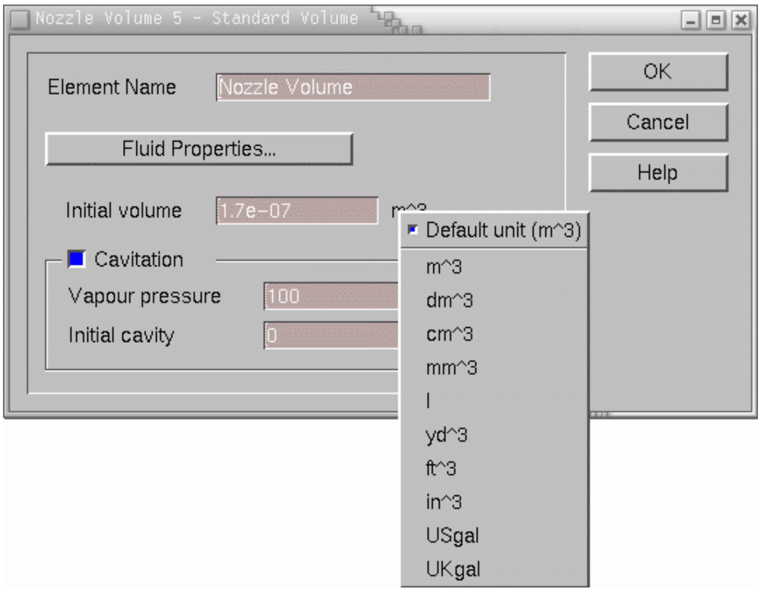


Figure 285: Unit Box of Volume Group

26.2.7. Utilities

The **Utilities** pull-down menu contains **Python Scripts** as shown in Figure 286:

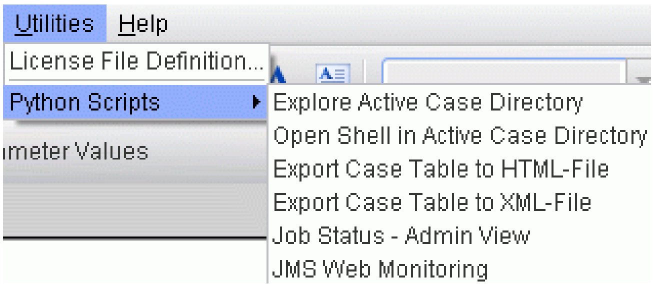


Figure 286: Utilities PullDownMenu with Python Scripts

The commands **Explore Active Case Directory** and **Open Shell in Active Case Directory** open the Explorer or Shell, respectively. The commands **Export Case Table to HTML-File** and **Export Case Table to XML-File** export the `Case Table` to the `html` or `xml` files, respectively (the file opens on the screen). As file name the BOOST Hydsim model file name is used, e.g. `common_rail.html`.

27. APPENDIX

Absolute(T)	Type:	Toggle Button
	Default:	Active
Additional mass (spring) **Additional moment of inertia**	Type:	Button
	Default:	Active
Ambient pressure	Type:	Real
	Unit:	Pressure Units
	Default:	1 bar
Ambient temperature	Type:	Real
	Unit:	Temperature Units
	Default:	293.15 K (20 °C)
Amperage (boost)	Type:	Real
	Unit:	Amperage Units
Amperage Table	Type:	Table (Real / Real)
	Unit:	Sim. Period Units / El. Current Units
	Default:	0. / 0.
Angle between cam and valve lever arms **Angle between cam-roller centerline/ support x-axis** **Angle between input/output lever arms** **Angle between perpendiculars of motion** **Angle between Ports 1 and 2** **Angle between pushrod motion/input arm** **Angle between valve side arm/valve motion** **Angle to x axis at calculation start**	Type:	Real
	Unit:	Angle Units
	Default:	0.
Angle of needle seat **Angle of needle tip**	Type:	Real
	Unit:	Angle Units
AREA TAB	Default: Hidden in background.	
Area table	Type:	Table (Real/Real/Real)
	Unit:	Length Units / Area Units / Area Units
	Default:	0. / 0./ 0.
Armature stroke (max. lift)	Type:	Real
	Unit:	Length Units

Automatic Reload	Type:	Button
	Default:	Not Active
Bearing Friction	Type:	Bar
	Default:	Active
Bearing friction force	Type:	Real
	Unit:	Force Units
	Default:	0.
Bearing (shaft) diameter	Type:	Real
	Unit:	Length Units
	Default:	0.
Bend angle	Type:	Real
	Unit:	Angle Units
	Default:	0.
Bend diameter **Bend radius**	Type:	Real
	Unit:	Length Units
	Default:	0.
Body area	Type:	Toggle Button
	Default:	Active
Body diameter	Type:	Toggle Button
	Default:	Not Active
Body stop damping at input end **Body stop damping at output end** **Body stop stiffness at input end** **Body stop stiffness at output end**	Type:	Real
	Unit:	Linear Stiffness Units
BTDC (firing)	Type:	Toggle Button
	Default:	Not active
Bubble radius at reference pressure	Type:	Real
	Unit:	Length Units
	Default:	0.05 μm
Bulk Modulus	Type:	Real
	Unit:	Stress Units
	Default:	1390 N/mm^2
Calculate bulk modulus internally from density-pressure function	Type:	Button
	Default:	Not Active
Calculate coef.	Type:	Bar
Calculation end time / End reference angle	Type:	Real
	Unit:	Simulation Period Units

	Default:	0.
Calculation Start	Type:	Toggle Button
	Default:	Active
Calculation start [angle BTDC]	Time:	Real
	Unit:	Angle Units
	Default:	0.
Cam plate acceleration at 1000 rpm	Type:	Toggle Button
	Default:	Active
Cam Plate Lift in normal direction (x)	Type:	Toggle Button
	Default:	Not Active
Cam Profile	Type:	bar
Cam profile lift in radial direction (x)	Type:	Toggle Button
	Default:	Not active
Cam profile table	Type:	Table (Real/Real)
	Unit:	Angle Units / Acceleration Units (Length Units)
	Default:	0. / 0.
Cam radial acceleration at 1000 rpm	Type:	Toggle Button
	Default:	Active
	Unit:	Angle Units / Length Units
	Default:	0. / 0.
Cam(shaft) mass	Type:	Real
	Unit:	Mass Units
	Default:	0.
Camshaft torsion Cavitation	Type:	Button
	Default:	Active
Charge duration	Type:	Real
	Unit:	Simulation Period Units
Clockwise	Type:	Toggle Button
	Default:	Not Active
Closed	Type:	Toggle Button
	Default:	Active
Coef.(Z>1) hydraulic	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Coef.(Z>1) cavitating	Type:	Real
	Unit:	Ratio Units

	Default:	1.
Common Rail with 2/2-Way Valve	Type:	Bar
COMPLIANT WALLS:	Type:	Button
	Default:	Not Active
Constant	Type:	Toggle Button
Constant	Type:	Toggle Button I
	Default:	Active
Constant	Type:	Toggle button II / III
	Default:	Active
Constant B **Constant C**	Type:	Real
	Unit:	Temperature Units
	Default:	0 K
Constant (basic model)	Type:	Button
	Default:	Active
CONSTANT CALCULATION STEP **CONSTANT FLUID PROPERTIES** **CONSTANT SOLID PROPERTIES**	Type:	Toggle button
	Default:	Active
CONSTANT FLOW RESISTANCE IN THROTTLE	Type:	Toggle Button II
	Default:	Active
Constant for laminar flow	Type:	Real
	Unit:	Ratio Units
	Default:	3500.
Constant for laminar flow C_lam	Type:	Real
	Unit:	Ratio Units
	Default:	455.
Constant/ Function of time	Type:	Toggle Button
	Default:	Not Active
Constant mass fraction	Type:	Toggle Button II
	Default:	Active
Constant solid properties	Type:	Toggle button
	Default:	Active
Contact damping ratio	Type:	Real
	Unit:	--
	Default:	0.1
Control	Type:	Bar
CONTROL FORCE	Type:	Table (Real / Real / ...)
	Unit:	Force Units

	Default:	0.
Corrector step (discrete time interval):	Type:	Integer
	Unit:	---
	Default:	1
Coulomb friction coefficient	Type:	Real
	Unit:	Ratio Units
	Default:	0.
Coulomb friction force **Coulomb friction force at translation** **Coulomb friction force in x-direction** **Coulomb friction force in y-direction**	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Force Units
	Default:	0.
Coulomb friction force	Type:	Button
	Default:	Not Active
	Type:	Real
	Unit:	Force Units
	Default:	0.
Coulomb friction torque	Type:	Button
	Default:	Active
Coulomb friction torque at rotation	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Torque Units
	Default:	0.
Counterclockwise	Type:	Toggle Button
	Default:	Active
Cross-section area	Type:	Toggle Button Toggle Button **I**
	Default:	Active
Cross-section diameter	Type:	Toggle Button Toggle Button **I**
	Default:	Not active
Cross-section diameter at input end **Cross-section diameter at output end**	Type:	Real
	Unit:	Length Units
	Default:	0.
Cross-section of plunger groove/bore: Narrowest area	Type:	Real
	Unit:	Area Units

Cross-section table	Type:	Table (Real / Real)
	Unit:	Length Units / Length Units
Cross-section type: **Circular** **Hollow**	Type:	Toggle Button
	Default:	Not active
Cross-section type: **Solid**	Type:	Toggle Button
	Default:	Active
Cross-sectional area at input end: **Cross-sectional area at output end:**	Type:	Real
	Unit:	Area Units
	Default:	0 Implies $A_{in} = A_{nom} = \frac{\pi D_h^2}{4}$.
Cross-sectional distance from input end:	Type:	Real / Real / Real / Real / Real / Real / Real / Real / Real / Real
	Unit:	Length Units
Cylinder length:	Type:	Real
	Unit:	Length Units
	Default:	0.
Cylindrical shape:	Type:	Toggle Button
	Default:	Active / Not Active
Damping **Damping coefficient**	Type:	Real
	Unit:	Linear Damping Units
	Default:	0.
Data File	Type:	Text
	Default:	-
Delay from Switch ON **Delay from Switch OFF**	Type:	Real
	Unit:	Time Units
	Default:	0.
Density	Type:	Real
	Unit:	Density Units
	Default:	7800 kg/m3
Depth	Type:	Real
	Unit:	Length Units
Depth at leading edge	Type:	Real
	Unit:	Length Units
Diameter **Diameter of valve ball**	Type:	Real
	Unit:	Length Units

Diameter at input end **Diameter at input throttle** **Diameter at output end** **Diameter at output throttle**	Default:	0.
Diametral gap at overlap **Diameter of one spray hole** **Diameter of needle collar** **Diameter of needle guide** **Diameter of needle seat** **Diameter of needle tip** **Diameter of needle collar** **Diameter of nozzle sac** **Diameter of upper guide bore** **Diametral gap at overlap**	Type:	Real
	Unit:	Length Units
Direction: x/y-axis (flow)	Type:	Button
Disable reverse flow (from output to input)	Type:	Button
	Default:	Not active
Discharge coef. for backward flow **Discharge coef. for forward flow**	Type:	Real
	Unit:	Dimensionless
	Default:	1.
Discharge coefficient at hole inlet	Type:	Button
	Default:	Not active
	Type:	Real
	Unit:	Ratio Units
Discharge Coefficient Table	Type:	Bar
	Type:	Table (Real / Real)
	Unit:	Ratio Units / Ratio Units
	Default:	1. / 0.7
Discharge duration	Type:	Real
	Unit:	Simulation Period Units
DISTANCE FROM PISTON CENTER OF GRAVITY **to roller end** **to base circle**	Type:	Real
	Unit:	Length Units
	Default:	0.
Distributed (advanced model)	Type:	Button
	Default:	Not active
Droplet size calculation	Type:	Pull Down Menu

Eccentricity of gravity center	Type:	Real
	Unit:	Length Units
	Default:	0.
Effective Cross-sectional Flow Area	Type:	Toggle Button II
	Default:	Active
Effective Flow Area (my*A)	Type:	Table (Real / Real)
	Unit:	Length Units / Area Units
	Default:	0. / 0.
Effective roller (contact) width **Effective width**	Type:	Real
	Unit:	Length Units
	Default:	0.
Elastic Body	Type:	Toggle Button
	Default:	Not Active
Elastic Camshaft	Type:	bar
Elastic rocker support **Elastic support** **Elasticity of lever arms** **Elasticity of rocker arm**	Type:	Button
	Default:	Active
Elastic Shaft (torsion)	Type:	Toggle Button
	Default:	Not Active
Electric capacitance	Type:	Real
	Unit:	Capacitance Units
Electromechanical transformer ratio	Type:	Real
	Unit:	Electromech. Transformer Ratio Units
Element	Type:	Pop-up menu
Element name	Type:	Text (up to 12 characters recommended)
	Default:	Rocker Arm
	Arbitrary text is specified to enable easier identification of the element. It is stored as a headline (up to the first 12 characters) in the corresponding output files. With `.ppd` file (Post-processor **Impress Chart Results | Model Tree**), the full name is used. With GIDas file (**Impress Chart Results | Load GIDas File**), up to 12 first characters are used (including spaces). With DATa file, up to first 8 characters will be used.	
ENGINE SPEED	Type:	Button
	Default:	Active

Engine Speed	Type:	Real
	Unit:	Angular Velocity Units
	Default:	2000 rpm
Engine type	Type:	Toggle Button
	Default:	4-Stroke
Equivalent torsional damping ratio **Equivalent viscous damping ratio**	Type:	Real
	Unit:	--
	Default:	1
Expansions / Contractions at Connections:	Type:	Button
	Default:	Not Active
Exponent for air gap **Exponent for amperage**	Type:	Real
Exponent of pressure-pulse damping:	Type:	Real
	Unit:	Ratio Units
	Default:	0.
Exponent of Reynolds number	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Exponent of Reynolds number n	Type:	Real
	Unit:	Ratio Units
	Default:	1.38
Export solution map	Type:	Button
	Default:	Not active
External Table	Type:	Real/Real
	Unit:	Sim. Period Units/not defined (values in SI Units)
Factor for 1st column **Factor for 2nd column (my)**	Type:	Button
	Default:	Active
	Type:	Real
	Default:	1.
Fill helix angle	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Angle Units
	Default:	0.
FILLING PORT	Type:	Button
	Default:	Active

Fill / spill port center	Type:	Real
	Unit:	Angle Units
Finger Support Friction	Type:	bar
Fixed Step	Type:	Toggle button
	Default:	Not Active when **Solver Options** is activated
FLOW AREA OR RATE TABLE	Type:	Table (Real / Real)
	Unit:	Length Units / Area Units (Volume Flow Units)
	Default:	0. / 0.
Flow Coefficient	Type:	Toggle button **II**
	Default:	Active
FLOW DISCHARGE COEFFICIENT (my)	Type:	Toggle Button
	Default:	Not active
Flow discharge coefficient (0<my<1)	Type:	Real
	Unit:	Ratio Units
	Default:	0.7
Flow discharge coefficient **Flow discharge coefficient away from edges**	Type:	Real
	Unit:	Ratio Units
	Default:	0.9
Flow discharge coefficient at leading edge	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Ratio Units
	Default:	0.9
Flow discharge coefficient at trailing edge	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Ratio Units
	Default:	0.7
Flow Rate	Type:	Toggle button **III** / **II**
	Default:	Not Active
Flow rate cavitating **Flow rate hydraulic** **Flow rate through throttle**	Type:	Real
	Unit:	Volume Flow Units
	Default:	0 m^3/s
Flow Rate vs. Time/Reference Angle:	Type:	Table (Real/Real)
	Unit:	Simulation Period Units / Vol. Flow Units

	Default:	0. / 0.
Flow Resistance	Type:	Bar
Flow Resistance coefficient	Type:	Toggle button III / II
	Default:	Active
Flow resistance coefficient (Z>1)	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Flow resistance coefficient at input end **Flow resistance coefficient at input throttle**	Type:	Real
	Unit:	Ratio Units
	Default:	0.5
Flow resistance coefficient at output end **Flow resistance coefficient at output throttle**	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Flow resistance coefficient at valve seat	Type:	Real
	Unit:	Ratio Units
	Default:	2.04
FLOW THROUGH SPRAY HOLES **FLOW THROUGH NEEDLE SEAT**	Type:	Button
	Default:	Active
Fluid damping at valve closing	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Linear Damping Units
	Default:	0.
Fluid-Gas Mixture (User defined gas fraction)	Type:	Toggle Button I
	Default:	Not Active
Fluid name	Type:	Combo box
FLUID PROPERTIES FROM PROPERTY DATABASE	Type:	Toggle Button
	Default:	Not active
Fluid to solid wall	Type:	Real
	Unit:	Heat transfer coefficient Units
Follower acceleration at 1000 rpm	Type:	Toggle Button
	Default:	Active
Follower lift at calculation start	Type:	Real
	Unit:	Length Units
	Default:	0.

Force Correction Factor	Type:	Bar
Force Correction Factor Table (SWITCH ON)	Type:	Table (Real / Real)
	Unit:	Sim. Period Units / Ratio Units
	Default:	0. / 1.
Force Correction Factor Table (SWITCH OFF)	Type:	Table (Real / Real)
	Unit:	Sim. Period Units / Ratio Units
	Default:	0. / 0.
Force magnitude (maximum)	Type:	Real
	Unit:	Force Units
Friction force	Type:	Real
	Unit:	Force Units
	Default:	0.
Friction force at input end **Friction force at output end**	Type:	Real
	Unit:	Force Units
Friction losses	Type:	Bar
Friction parameters	Type:	Button
Friction torque at input end **Friction torque at output end**	Type:	Real
	Unit:	Torque Units
Function of gap and time	Type:	Toggle Button
	Default:	Active
Function of Voltage **Further Armature (Valve Body) Data**	Type:	Bar
Gain (d33)	Type:	Real
	Unit:	Electric Field Strain Units
GAP LENGTH	Type:	Toggle Button I
Gas bubble dynamics	Type:	Toggle Button I
	Default:	Active
Gas constant R	Type:	Real
	Unit:	Gas Constant Units
	Default:	287 J/kgK
Gas Density	Type:	Bar
Geometric and Material Data …	Type:	Bar
GLOBAL SOLID PROPERTIES	Type:	Toggle Button
	Default:	Active

Grinder phase-out line	Type:	Button
	Default:	Not Active
	Type:	Real
	Unit:	Angle Units
Grinding inclination angle	Type:	Real
	Unit:	Angle Units
Groove depth	Type:	Real
	Unit:	Length Units
Half-angle of valve seat	Type:	Real
	Unit:	Angle Units
Head Grinding Data … **Head Groove Data …**	Type:	Bar
Heat transfer coef.	Type:	Real
	Unit:	Heat transfer coefficient Units
Heat transfer to ambience **Heat transfer through Walls:**	Type:	Button
	Default:	Not Active
Helix angle: First/Second	Type:	Real
	Unit:	Angle Units
	Default:	0./0.
HERTZ STRESS CALCULATION	Type:	Button
	Default:	Active
Hole discharge coefficient (at cavitation)	Type:	Real
	Unit:	Ratio Units
	Default:	0.65
Hole discharge coefficient (no cavitation)	Type:	Real
	Unit:	Ratio Units
	Default:	0.7
<u>Hole inlet radius</u> **Hole outlet diameter**	Type:	Real
	Unit:	Length Units
HOLLOW NEEDLE	Type:	Button
	Default:	Not Active
Hydraulic diameter:	Type:	Real
	Unit:	Length units
	Default:	0.
HYSTERESIS (of one ceramic layer)	Type:	Button
	Default:	Active

Impact factor	Type:	Real
	Unit:	---
	Default:	1.
Inbowl swirl	Type:	Real
	Unit:	Ratio Units
	Default:	0.
Increment S1 (input) **Increment S2 (output)**	Type:	Real
	Unit:	Length Units
	Default:	0.
Inertia moment at input end **Inertia moment at output end**	Type:	Real
	Unit:	Moment of Inertia Units
Initial angle to filling port	Type:	Real
	Unit:	Angle Units
Initial cavity	Type:	Real
	Units:	Volume Units
	Default:	0.
Initial collar overlap	Type:	Real
	Unit:	Length Units
Inner diameter:	Type:	Real
	Unit:	Length Units
	Default:	0.
Initial follower velocity at 1000 rpm	Type:	Real
	Unit:	Velocity Units
	Default:	0.
Initial gap length	Type:	Real
	Unit:	Length Units
	Default:	0.
Initial plate velocity at 1000 rpm	Type:	Real
	Unit:	Velocity Units
	Default:	0.
Initial pressure p0	Type:	Real
	Unit:	Pressure Units
	Default:	100000 Pa (1 bar)
Initial temperature T0	Type:	Real
	Unit:	Temperature Units
	Default:	293.15 K (20 °C)

Initial volume	Type:	Real
	Unit:	Volume Units
	Default:	0.
Input Vector Name (optional)	Type:	Button
	Default:	Not Active
	Type:	Text (up to 12 characters recommended)
	Default:	-
<u>Input voltage table</u>	Type:	Table (Real/Real)
	Unit:	Simulation Period Units/Voltage Units
	Default:	0./0.
Insert VIRTUAL VOLUMES	Type:	Toggle button
	Default:	Active
Invert CONNECTIONS	Type:	Toggle button
	Default:	Not Active
Isothermal flow (weakly compressible)	Type:	Toggle button
	Default:	Active
k-factor	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Ratio Units
Kinematic viscosity	Type:	Real
	Unit:	Kinematics Viscosity Units
	Default:	3 m^2/s
Layer mass	Type:	Real
	Unit:	Mass Units
Leading edge of grinding	Type:	Real
	Unit:	Angle Units
Length of cam side lever arm (support-roller) **Length of input side lever arm (pushrod) / (cam)** **Length of output side lever arm (valve)** **Length of valve side lever arm (support-finger)**	Type:	Real
	Unit:	Length Units
	Default:	0.
Length of one spray hole	Type:	Real
	Unit:	Length Units

Lift of reciprocating cam follower	Type:	Toggle Button
	Default:	Not Active
Line length:	Type:	Real
	Unit:	Length Units
	Default:	0.
Lower bound (SI)	Not implemented yet	
Magnetic Force Calculation Parameters	Type:	Bar
Magnetic force constant	Type:	Real
	Unit:	-- Dimension will be generated according to the dimensions of magnetic force and air gap and according to the values of amperage and air gap exponents
Magnetic Force vs. Lift and Amperage	Type:	Bar
Mass	Type:	Real
	Unit:	Mass Units
	Default:	0.
Mass fraction of gas	Type:	Real
	Unit:	Ratio Units
	Default:	0.
Max. idle stroke of one layer	Type:	Real
	Unit:	Length Units
	Default:	0.
Max. Force Table	Type:	Table (Real / Real)
	Unit:	Length Units / Force Units
	Default:	0. / 0.
Max.Step Size	Type:	Real
	Unit:	Time Units
	Default:	0.
Maximum body lift **Maximum primary lift** **Maximum total lift**	Type:	Real
	Unit:	Length Units
Maximum lift of valve body **Maximum lift of valve ball** **Maximum lift of needle**	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Length Units
Maxwell resistive Capacitor (MRC) elements	Type:	Toggle Button

	Default:	Not Active
Mechanical damping **Mechanical stiffness**	Type:	Real
	Unit:	Linear Damping Units
Mechanical transmision ratio	Type:	Real
	Default:	1.
Min. air gap **(to magnet pole)** **(armature-pole)**	Type:	Real
	Unit:	Length Units
Moment of inertia	Type:	Real
	Unit:	Mass Units
	Default:	0.
Moment of inertia about rotation axis	Type:	Real
	Unit:	Moment of Inertia Units
	Default:	0.
Moving mass	Type:	Real
	Unit:	Mass Units
	Default:	0.
MRC element table	Type:	Table (Real/Real)
	Unit:	Electric Stiffness Units / Voltage Units
	Default:	0./0.
my*A of valve seat at opening **my*A of valve seat at closing**	Type:	Table (Real / Real)
	Unit:	Sim. Period Units / Area Units
	Default:	0. / 0.
Needle lift at unthrottled seat	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Length Unit
	Default:	0.
<u>Needle Lift-dependent</u>	Type:	Toggle Button I
	Default:	Not Active
	Type:	Table (Real / Real / Real)
	Unit:	Length Units / Ratio Units / Ratio Units
Needle lift tolerance open/closed)	Type:	Real
	Unit:	Length Units
	Default:	10 μm

Needle seat damping **Needle seat stiffness** **Needle stop damping** **Needle stop stiffness**	Type:	Real
	Unit:	Linear Stiffness Units
Negative flow resistance coefficient at input end:	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Negative flow resistance coefficient at output end:	Type:	Real
	Unit:	Ratio Units
	Default:	0.5
Normlized Stroke-Voltage diagram (idle)	Type:	Toggle Button
	Default:	Active
Nozzle diameter at spray holes	Type:	Real
	Unit:	Length Units
Nozzle rate at max. needle lift	Type:	Real
	Unit:	Volume Flow Units
	Default:	0 m^3/s
Number of ceramics layers	Type:	Integer
	Default:	1.
Number of line cells:	Type:	Button
	Default:	Active
	Type:	Integer
	Default:	---
Number of pistons	Type:	Integer
	Default:	4 / 1
Number of ports **Number of spill ports**	Type:	Integer
	Default:	1
Number of positions S1 (input) **Number of positions S2 (output)**	Type:	Integer
	Default:	0
Number of roots of Bessel function of order zero:	Type:	Integer
	Unit:	---
	Default:	20
Number of spray holes	Type:	Integer
Number of values to store	Type:	Integer
	Unit:	--
	Default:	100

Open	Type:	Toggle Button
	Default:	Not active
Opening pressure **Opening /closing pressure (+/-)**	Type:	Real
	Unit:	Pressure Units
Orifice type:	Type:	Toggle Button
	Default:	Standard
Other	Type:	Toggle Button
	Default:	Active
Outer surface area	Type:	Real
	Unit:	Area Units
Output	Type:	Toggle button
	Options:	GIDas/DATa/GIDas+DATa
	Default:	GIDas
Output domain	Type:	Toggle Button
	Default:	Ref. Angle
Output Vector Name (optional)	Type:	Button
	Default:	Not Active
	Type:	Text (up to 12 characters recommended)
	Default:	-
Parameter (table)	Type:	Real
	Unit	Not defined (values in SI Units)
	Default:	0.
Penetration Calculation	Type:	Pull Down Menu
<u>Piezo Gain</u>	Type:	Toggle button I
	Default:	Active
Piezo Strain along Polarization (d33)	Type:	Table (Real/Real)
	Unit:	Voltage Units/ Electric Field Strain Units
	Default:	0./0.
Piston area at input end **Piston area at output end**	Type:	Real
	Unit:	Area Units
	Default:	0.
Piston diameter **Piston diameter at input end** **Piston diameter at output end**	Type:	Button

	Default:	Active
	Type:	Real
	Unit:	Length Units
	Default:	0.
Piston seat damping	Type:	Real
	Unit:	Linear Damping Units
	Default:	0.
Piston seat diameter	Type:	Real
	Unit:	Length Units
	Default:	0.
Piston seat stiffness	Type:	Real
	Unit:	Linear Stiffness Units
	Default:	0.
Piston stop area at input end **Piston stop area at output end**	Type:	Real
	Unit:	Area Units
	Default:	0.
Piston stop cross-section	Type:	Button
	Default:	Not active
Piston stop damping **Piston stop damping at input end** **Piston stop damping at output end**	Type:	Real
	Unit:	Linear Damping Units
	Default:	0.
Piston stop diameter at input end **Piston stop diameter at output end**	Type:	Real
	Unit:	Length Units
	Default:	0.
Piston stop stiffness **Piston stop stiffness at input end** **Piston stop stiffness at output end**	Type:	Real
	Unit:	Linear Stiffness Units
	Default:	0.
Piston stroke (max. lift)	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Length Units
	Default:	0.
Plate lift at calculation start	Type:	Real
	Unit:	Length Units
	Default:	0.
Plunger diameter	Type:	Button
	Default:	Not Active

	Type:	Real
	Unit:	Length Units
	Default:	0.
Plunger slot width	Type:	Real
	Unit:	Length Units
Poisson's ratio **Poisson's ratio of cam material**	Type:	Real
	Unit:	--
	Default:	0.3
Poisson's ratio of pump material	Type:	Real
	Unit:	Ratio Units
	Default:	0.3
Port diameter **Port height size** **Port width size**	Type:	Real
	Default:	0.
	Units:	Length Units
Positive flow resistance coefficient at input end:	Type:	Real
	Unit:	Ratio Units
	Default:	0.5
Positive flow resistance coefficient at output end:	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Precharging voltage	Type:	Button
	Default:	Not Active
	Type:	Real
	Unit:	Voltage Units
	Default:	0.
PREDICTOR - CORRECTOR MODEL:	Type:	Toggle Button
	Default:	Active
PREDICTOR MODEL (EMPIRICAL damping):	Type:	Toggle Button
	Default:	Not active
Preload	Type:	Real
	Unit:	Force Units
	Default:	0.
Preload force	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Force Units
	Default:	0.

Pressure:	Unit:	Pressure Units
	Default:	100000 Pa (1 bar)
Pressure and Temperature vs. Time/Reference Angle:	Type:	Table (Real/Real)
	Unit:	Simulation Period Units / Pressure Units / Temperature Units
	Default:	0. / 1 bar / 20 degC
Pressure difference	Type:	Real
	Unit:	Pressure Units
	Default:	100 bar
Pressure – Flow Characteristic Table	Type:	Table (Real / Real)
	Unit:	Volume Flow Units / Pressure Units
	Default:	0. / 0.
Pressure in cam chamber **Pressure input-output** **Pressure input-vapour**	Type:	Real
	Unit:	Pressure Units
	Default:	0 bar
Program Calculated **Program-calculated heat convection coef.** **Program-calculated flow resistance of port**	Type:	Toggle Button
	Default:	Active
Program-calculated annular gap (compliant volume)	Type:	Toggle Button II
	Default:	Not Active
PROGRAM-CALCULATED FLOW AREA (standard geometry**)**	Type:	Toggle Button I
	Default:	Active
Program-calculated flow discharge coefficient	Type:	Toggle Button
	Default:	Not active
Property	Type:	Pop-up menu
Radial clearance	Type:	Real
	Unit:	Length Units
Radius **Radius of cam base circle** **Radius of cam scrolring** **Radius of cam plate** **Radius of roller**	Type:	Real
	Unit:	Length Units
	Default:	0.
Radius of throttle tip	Type:	Real
	Unit:	Length Units

Rectangular shape	Type:	Toggle Button
	Default:	Active
Reference density	Type:	Real
	Unit:	Density Units
	Default:	820 kg/m^3
Reference pressure	Type:	Real
	Unit:	Pressure Units
	Default:	100.000 Pa (1 bar)
Reference pressure in fluid	Type:	Real
	Unit:	Pressure Units
	Default:	1 Pa
Reference Speed	Type:	Real
	Unit:	Angular Velocity Units
	Default:	1000 rpm
Reference temperature	Type:	Real
	Unit:	Temperature Units
	Default:	293.15 K (20 degC)
Relative(delta T)	Type:	Toggle Button
	Default:	Not active
Relief grinding	Type:	Toggle Button
	Default:	Not Active
Residual induction constant	Type:	Real
	Unit:	Dimension will be generated according to the dimension of air gap and values of air gap exponents
Response Time	Type:	Real
	Unit	Time Units
	Default:	0.
Retraction (unloading) volume	Type:	Real
	Unit:	Volume Units
Rigid Body	Type:	Toggle Button
	Default:	Active
Rigid mass at input end **Rigid mass at output end**	Type:	Real
	Unit:	Mass Units
Rigid shaft	Type:	Toggle Button
	Default:	Active

Rocker arm mass	Type:	Real
	Unit:	Mass Units
	Default:	0.
Rocker bearing friction	Type:	bar
Sampling Time	Type:	Real
	Unit	Time Units
	Default:	0.
Save interval for restart	Type:	Button
	Default:	Not active
	Type:	Integer
	Unit:	-----
	Default:	0
Scaling factor for 1st column: **Scaling factor for 2nd column (my)**	Type:	Button
	Default:	Active
	Type:	Real
	Default:	1.
Seat length	Type:	Real
	Unit:	Length Units
Shift angle relative to calculation start	Type:	Real
	Unit	Angle Units
	Default:	0.
Shift Time/Angle relative to calculation start	Type:	Real
	Unit	Simulation Period Units
	Default:	0.
Shoe radius	Type:	Real
	Unit:	Length Units
	Default:	0.
Sholdered/ tapered **Sholdered Head**	Type:	Toggle Button
	Default:	Active
Show	Type:	Bar
Simulation Domain:	Type:	Toggle Button
	Default:	Time
Simulink Model Name	Type:	Text (up to 462 characters)
Simulink Model State	Type:	Combo box
	Default:	Continuous
Solid name	Type:	Combo box
Solid Properties …	Type:	Bar

SOLID PROPERTIES FROM PROPERTY DATABASE	Type:	Toggle Button
	Default:	Not Active
Solver	Type	Combo box
	Default	ode45 (Domand-Prince)
Solver	Type	Combo box
	Default	discrete (no continuous states)
Solver Options (optional)	Type:	Button
	Default:	Not Active
Special Head Geometry	Type:	Button
	Default:	Not Active
Specific heat	Type:	Real
	Unit:	Specific Heat Units
	Default:	1960 J/kgK
Specific heat capacity	Type:	Real
	Unit:	Specific Heat Units
	Default:	452 J/kgK
Specific heat ratio k	Type:	Real
	Unit:	Ratio Units
	Default:	1.4
Spherical shape:	Type:	Toggle Button
	Default:	Not Active
Spray Calculation	Type:	Bar
Spray cone angle calculation	Type:	Pull Down Menu
	Default:	(none)
Spray hole length	Type:	Real
	Unit:	Length Units
SQUEEZING FLUID AT NEEDLE CLOSING	Type:	Button
	Default:	Not Active
Start / End table	Type:	Table (Real / Real)
	Unit:	Sim. Period Units / Sim. Period Units
	Default:	0. / 0.
Starting value S1 (input) **Starting value S2 (output)**	Type:	Real
	Unit:	Length Units
	Default:	0.
Step Size	Type:	Real
	Unit:	Time Units

	Default:	0.
Stiffness	Type:	Real
	Unit:	Linear Stiffness Units
	Default:	0.
Stiffness normal to input side arm (pushrod) / (cam) **Stiffness normal to output side arm (valve)**	Type:	Real
	Unit:	Linear Damping Units
	Default:	0.
Stiffness of serial spring to piston	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Linear Stiffness Units
	Default:	0.
Stop Area	Type:	Toggle Button
	Default:	Active
Stop Diameter	Type:	Toggle Button
	Default:	Not active
Stroke: Pre-stroke/ Effective stroke	Type:	Real
	Unit:	Length Units
	Default:	0./0.
Stroke-Voltage table	Type:	Table (Real/Real)
	Unit:	Voltage Units/Ratio Units
	Default:	0./0.
Support bearing stiffness **Support bearing damping**	Type:	Real
	Unit:	Linear Stiffness Units
	Default:	0.
Suppress runtime output in Logfile	Type:	Button
	Default:	Active
Surface tension	Type:	Real
	Unit:	Linear Stiffness Units
	Default:	0.029 N/m
Switched off	Type:	Toggle Button
	Default:	Active
Switched on	Type:	Toggle Button
	Default:	Not Active
TABLE	Type:	Table (Real / Real)
	Unit:	Simulation Period Units / SI Units
	Default:	0. / 0.

Table	Type:	Real / Real / Real / Real / Real
	Unit:	Pressure Units **/** Stress Units **/** Density Units **/** Kinematic Viscosity Units / Linear Stiffness Units
	Default:	0. / 0. / 0. / 0. / 0.
Table	Type:	Bar
Table	Type:	Real / Real / Integer
	Unit:	Simulation Period Units **/** Simulation Period Units **/** **--**
	Default:	0. / 0. / 0.
Table 1 – 2	Type:	Bar
Table 1 – 2	Type:	Table (Real / Real)
	Unit:	Ratio Units / Ratio Units
	Default:	0. / 0.
Table of Boundary Conditions:	Type:	Table (Real/Real/Real/Real)
	Default:	0. / 0. / 0. / 0.
Tapered Head	Type:	Toggle Button
	Default:	Not Active
Taper angle	Type:	Real
	Unit:	Angle Units
Target function: Maximum	Type:	Toggle button
Target function: Minimum	Default:	Not implemented yet
Target function: Value	Type:	Toggle button
	Default:	Active
Temperature	Type:	Real
	Unit:	Temperature Units
	Default:	293.15 K (20 °C)
Termination error [%]	Type:	Toggle button
	Default:	Active
Thermal conductivity	Type:	Real
	Unit:	Heat Conductivity Units
	Default:	50 W/(mK)
Thermal conductivity	Type:	Real
	Unit:	Heat Conductivity Units
	Default:	0.145 W/mK

Thermal expansion coef.	Type:	Real
	Unit:	Thermal Expansion Units
	Default:	0.001 1/K
Thermal flow (weakly compressible)	Type:	Toggle button
	Default:	Not Active
Throttle cross-sectional area	Type:	Real
	Unit:	Area Units
Throttle cross-section diameter **Throttle diameter**	Type:	Real
	Unit:	Length Units
<u>Time/Crank Angle</u>	Type:	Table (Real / Real)
	Unit:	Sim. Period Units / Sim. Period Units
	Default:	0. / 0.
Time function (one switching event)	Type:	Toggle Button
	Default:	Active
Time step / Reference angle step	Type:	Real
	Unit:	Simulation Period Units
	Default:	1.e-7 s
Tolerance (original units)	Type:	Toggle button
	Default:	Not active
Tolerance (SI)	Type:	Check box
	Default:	Active
Torsional damping	Type:	Real
	Unit:	Rotational Damping Units
	Default:	0.
Torsional stiffness	Type:	Real
	Unit:	Torsional Stiffness Units
	Default:	0.
Trailing edge angle	Type:	Button
	Default:	Not Active
	Type:	Real
	Unit:	Angle Units
Trailing edge of grinding	Type:	Real
	Unit:	Angle Units
<u>Transformer ratio</u>	Type:	Toggle button I
	Default:	Not Active
Tube cross-sectional area	Type:	Real
	Unit:	Area Units

Tube cross-section diameter **Tube diameter**	Type:	Real
	Unit:	Length Units
Turbulent flow coefficient	Type:	Bar
Turbulent flow constant at needle seat	Type:	Real
	Unit:	Ratio Units
	Default:	1.
Turbulent flow constant C_turb	Type:	Real
	Unit:	Ratio Units
	Default:	1.2
Type	Type:	Pop-up menu
Type of nozzle + ID number + "element name"	Type:	Button
	Default:	Not active
Type of Property	Type:	Pop-up menu
Unit Injector with SID Piston	Type:	Bar
Upper bound (SI)	Not implemented yet	
Use Default Data File Path	Type:	Button
	Default:	Not Active
User Defined **<u>User-defined (3D-table)</u>** **USER-DEFINED ANIMATION RANGE** **User-defined heat convection coef.** **User-defined Range (optional)**	Type:	Toggle Button
	Default:	Not Active
User-defined annular gap (standard volume)	Type:	Toggle Button II
	Default:	Active
	Type:	Table (Real / Real)
	Unit:	Pressure Units / Length Units
	Default:	0. / 0.
USER-DEFINED FLOW AREA OR RATE (special collar geometry) **USER-DEFINED FLOW AREA (special collar geometry)**	Type:	Toggle Button I
	Default:	Not Active
<u>User-defined flow discharge coefficient</u>	Type:	Toggle Button
	Default:	Active
User-defined flow resistance **User-defined flow resistance of port**	Type:	Toggle Button
	Default:	Not active
	Type:	Table (Real/Real)
	Unit:	Area Units / Ratio Units
	Default:	0. / 0.

User-defined Range Table	Type:	Table (Real / Real)
	Unit:	Simulation period Units / Simulation period Units
	Default:	0./0.
Value	Type:	Real
	Unit:	Not defined (values in SI Units)
	Default:	-
Valve seat diameter	Type:	Real
	Unit:	Length Units
Valve seat damping **Valve seat stiffness** **Valve spring damping** **Valve spring stiffness** **Valve stop damping** **Valve stop stiffness**	Type:	Real
	Unit:	Linear Stiffness Units
Vapor pressure	Type:	Real
	Unit:	Pressure Units
	Default:	100 Pa
Variable	Type:	Toggle button II / III
	Default:	Not Active
Variable:	Type:	Toggle Button
	Default:	*x*-coordinate
VARIABLE CALCULATION STEP **Variable cylinder charge properties**	Type:	Toggle Button
	Default:	Not active
VARIABLE FLOW RESISTANCE IN THROTTLE	Type:	Toggle Button II
	Default:	Not Active
Variable flow resistance table	Type:	Table (Real / Real)
	Unit:	Ratio Units / Ratio Units
	Default:	0. / 0.
VARIABLE FLUID PROPERTIES	Type:	Toggle Button
	Default:	Not active
Variable Step	Type:	Toggle button
	Default:	Active when **Solver Options** is activated
Vector Name	Type:	Text (up to 14 characters recommended)
	Default:	User_vector
VISCOSITY (VOGEL)	Type:	Bar

Viscous friction coefficient	Type:	Real
	Unit:	Linear Damping Units
	Default:	0.
Void fraction at reference pressure	Type:	Real
	Unit:	Ratio Units
	Default:	0.25 %
Volumetric Flow (Nozzle Rate) **Volumetric void fraction**	Type:	Toggle Button II
	Default:	Not Active
Volumetric void fraction table	Type:	Table (Real / Real)
	Unit:	Pressure Units / Ratio Units
	Default:	0. / 0.
WALL DEFORMATION:	Type:	Button
	Default:	Active
Wall thickness:	Type:	Button
	Default:	Active
	Type:	Real
	Unit:	Length Units
	Default:	0.
Wall to ambience	Type:	Real
	Unit:	Heat transfer coefficient Units
Width	Type:	Real
	Unit:	Length Units
Young's modulus: (LINE)	Type:	Real
	Unit:	Pressure Units
	Default:	0.
Young's modulus **Young's modulus (SOLID PROPERTIES)**	Type:	Real
	Unit:	Stress Units
	Default:	210000 N/mm^2
Young’s modulus of cam material **Young’s modulus of pump material**	Type:	Real
	Unit:	Pressure Units
	Default:	2.1e+11 N/m^2
#	Type:	Integer
	Default:	Serial number

Angle		
input w	Unit:	Angle Units
output w	Unit:	Angle Units
Angular Velocity		
input ω	Unit:	Velocity Units
output ω	Unit:	Velocity Units
angular velocity in w-direction	Unit:	Angular Velocity Units
velocity in x-direction	Unit:	Velocity Units
velocity in y-direction	Unit:	Velocity Units
w-angle	Unit:	Angle Units
x-coordinate	Unit:	Length Units
y-coordinate	Unit:	Length Units
X-coordinate		
input x	Unit:	Length Units
output x	Unit:	Length Units
Velocity in x-direction		
input v	Unit:	Velocity Units
output v	Unit:	Velocity Units

Initial Conditions

Angle	Unit:	Angle Units
Angular Velocity	Unit:	Angular Velocity Units
Flow Rate at x-input end:	Unit:	Volume Flow Units
Flow Rate at x-output end:	Unit:	Volume Flow Units
w-angle	Unit:	Angle Units
angular velocity in w-direction	Unit:	Angular Velocity Units