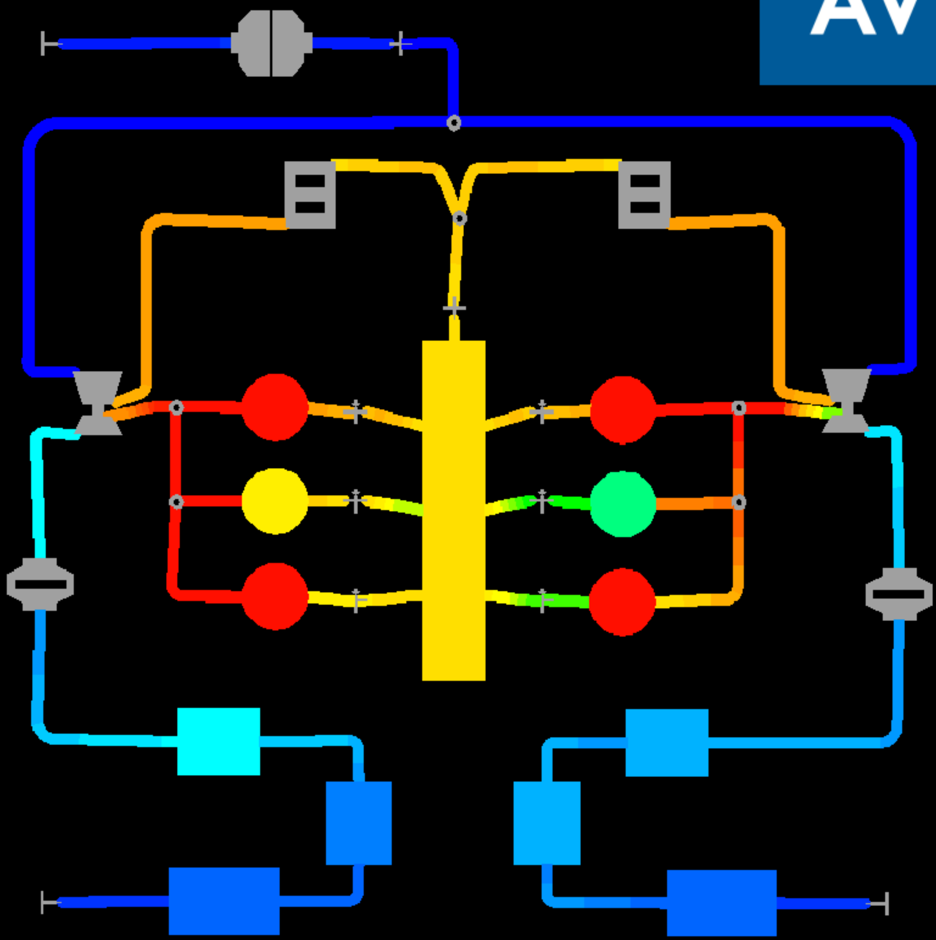


AVL BOOST™

Version 2018

4 Cylinder Gasoline Engine Example

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

AST Local Support Contact: www.avl.com/simulation-technologies

Revision	Date	Description	Document No.
A	17-Jul-2009	BOOST v2009 - Examples	01.0105.2009
B	30-Nov-2009	BOOST v2009.1 - Examples	01.0105.2009.1
C	19-Nov-2010	BOOST v2010 - Examples	01.0105.2010
D	25-Mar-2011	BOOST v2010.1 - Examples	01.0105.2010.1
E	29-Jul-2011	BOOST v2011 - Examples	01.0105.2011
F	02-Dec-2011	BOOST v2011.1 - Examples	01.0105.2011.1
G	16-May-2012	BOOST v2011.2 - Examples	01.0105.2011.2
H	20-Jun-2013	BOOST v2013.1 – 4 Cylinder Gasoline Example	01.0105.2013.1
I	16-Sept-2013	BOOST v2013.1 – 4 Cylinder Gasoline Example	01.0105.2013.1.1
J	15-Nov-2013	BOOST v2013.2 – 4 Cylinder Gasoline Example	01.0105.2013.1.2
K	28-Feb-2015	BOOST v2014.1 – 4 Cylinder Gasoline Example	01.0105.2014.1
L	26-Feb-2016	BOOST v2016 – 4 Cylinder Gasoline Example	01.0105.2016
M	30-Oct-2016	BOOST™ v2017 – 4 Cylinder Gasoline Example	01.0105.2017
N	26-May-2017	BOOST™ v2017.1 – 4 Cylinder Gasoline Example	01.0105.2017.1
O	02-Feb-2018	BOOST™ v2018 – 4 Cylinder Gasoline Example	01.0105.2018

This document describes how to run the BOOST™ software. It does not attempt to discuss all the concepts of 1D gas dynamics required to obtain successful solutions. It is the user's responsibility to determine if he/she has sufficient knowledge and understanding of gas dynamics to apply this software appropriately.

Contents

1. 4 CYLINDER GASOLINE ENGINE EXAMPLE

This chapter describes how to create and run the model of a 4 cylinder SI engine running at 5000 rpm WOT. The `ottocalc.bwf` file is used for this example and is located in the `spark_ignited` folder.

The `ottocalc_species.bwf` and `ottocalc_species_2zone.bwf` examples are available using the **General Species Transport** option. The setup is identical to the `ottocalc.bwf` except that the general species transport option is used, BMEP control is disabled and Monitor element is removed. General Species option is described in section 1.6.

1.1. Design the Model

The model can be designed by placing the elements in the working area first and then connecting them with the pipes. Alternatively elements can be placed in the required order. The following figure displays the created model:

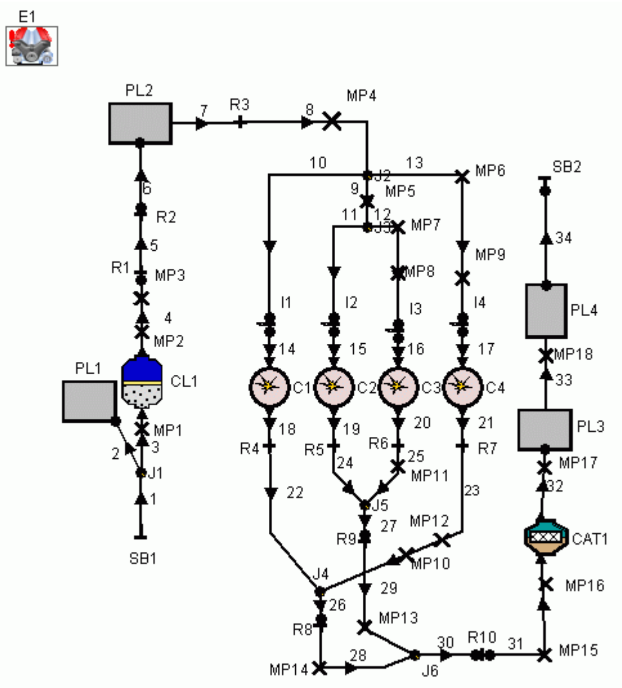


Figure 1-1: Model Schematic for Four Cylinder Engine Model

The model consists of the following elements:

- 4 Cylinders C
- 1 Engine E
- 1 Air Cleaner CL
- 1 Catalyst CAT
- 4 Injectors I
- 2 System Boundaries SB
- 4 Plenums PL
- 6 Junctions J
- 10 Restrictions R

- 18 Measuring Points	MP
- 34 Pipes	Numbers

Double-click the required element in the Element tree with the left mouse button to display it in the working area. Move the displayed element to the desired location with the left mouse button. Select  to insert a pipe and attach it to the required elements by clicking on the activated circles (triangles for the cylinder, air cleaner and cooler).

For the intake system the pipe and junction model is preferred to the plenum model. This is because the plenum model may predict equal air distribution which is often a critical issue especially for long receivers with small cross sectional areas. The step in cross sectional area at the inlet to the intake receiver is modeled with a flow restriction. Special attention has to be paid to correct modeling of the length of the intake runners as shown in the following figure.

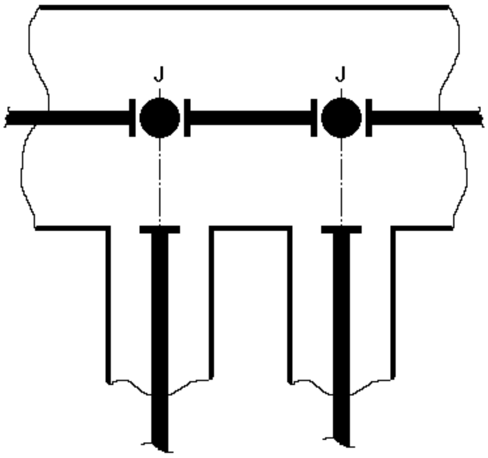


Figure 1-2: Modeling of an Intake Receiver with Pipes and Junctions

The following figure shows three different models for an intake receiver of a four cylinder engine:

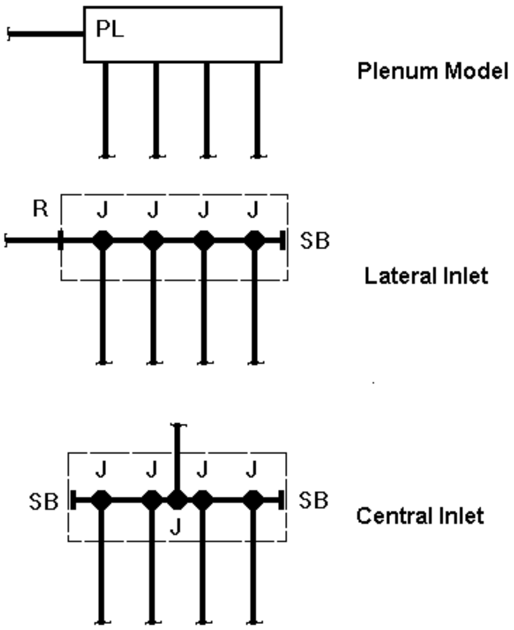


Figure 1-3: Intake Receiver Models

The first model is a simple plenum model. The second is a pipe and junction model with lateral inlet and the third is a pipe and junction model with central inlet. The total volume of the receiver was kept constant. Figure 1-4 shows the predicted volumetric efficiency and air distribution for the three models. The air distribution is expressed as the difference between the maximum and minimum volumetric efficiency of an individual cylinder related to the average volumetric efficiency.

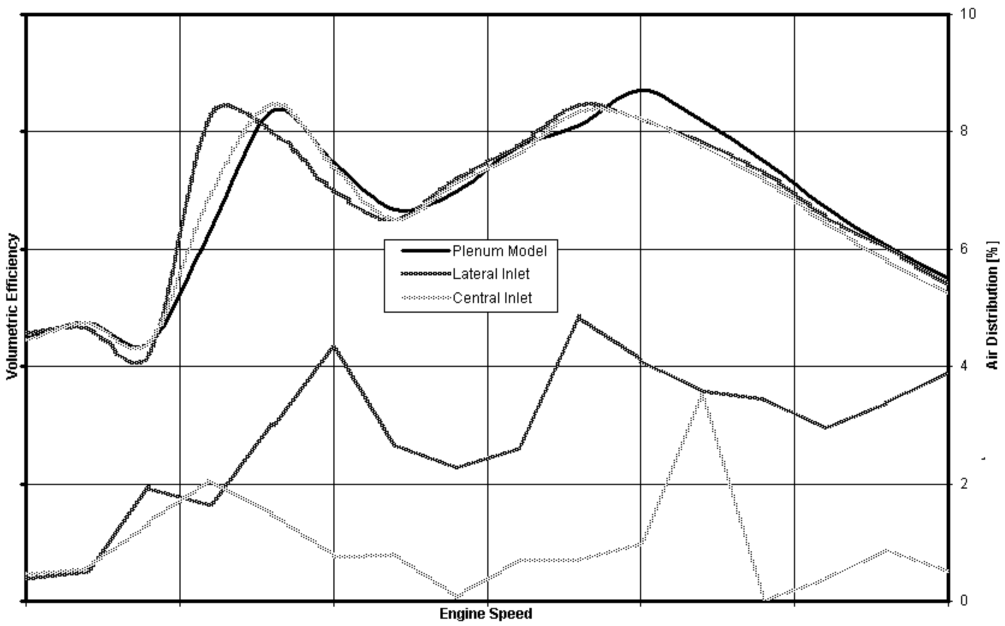


Figure 1-4: Influence of Intake Receiver Modeling on Volumetric Efficiency and Air Distribution

The predicted overall volumetric efficiency is similar for all three models, except for shifts in the resonance speeds. As the plenum model does not account for pressure waves in the intake receiver, equal volumetric efficiencies are calculated for all cylinders. The lateral air feed proves to be most critical with respect to air distribution especially at higher engine speeds.

1.2. General Input Data

BOOST requires the specification of the general input data prior to the input of any element.

The global input data must be defined first. Select **Simulation** | **Control** to access the input fields.

1. **SIMULATION TASKS**

   Click on the **Simulation Tasks** sub-group folder in the tree and select **Cycle Simulation**.

2. **CYCLE SIMULATION**

   Click on the **Cycle Simulation** sub-group folder in the tree and enter the following data:

Species Transport:	Classic (default)
Simulation Interval	
End of Simulation	50 cycles
Convergence Control	Activate
Spatial Pipe Discretization	
Average Cell Size	30 mm

NOTE

Non-Engine Application is deactivated when the Engine element is introduced.

3. **CLASSIC SPECIES SETUP**

   Click on the **Classic Species Setup** sub-group folder to access the input fields. The input values used are default.

4. **INITIALIZATION**

   Click on the **Initialization** sub-group folder. Select **A/F-Ratio** from the **Ratio** pull-down menu. Select **Add Set** and enter the data in the input fields for each set.

Set	Pressure bar	Temp degC	Fuel Vapour	Combustion Products	A/F Ratio
1	1	30.85	0	0	10000
2	0.95	66.85	0.07	0	10000
3	1.5	216.85	0	1	14.3

5. **CONVERGENCE CONTROL**

   Click on the **Convergence Control** sub-group folder and enter the following data:

   Click on **Add row** and click on the input field to show the list of options. Then select **Finish**.

	Element	Parameter	Value	Unit
1	Cylinder 1	IMEP	500	Pa
2	Cylinder 2	IMEP	500	Pa
3	Cylinder 3	IMEP	500	Pa
4	Cylinder 4	IMEP	500	Pa

6. **RESTART CONTROL**

   Click on the **Restart Control** sub-group folder to access the input fields. Select **Specific Interval** from the **Restart File Saving Interval** pull-down menu and enter `720` deg for **Saving Interval**.

7. **OUTPUT CONTROL**

   Click on the **Output Control** sub-group folder and enter `5` deg for **Saving Interval**.

Click **OK**.

1.3. Element Input Data

Select the displayed element with the right mouse button and select **Properties** from the submenu to open the relevant data input window. Alternatively double click on the element with the right mouse button.

Data can be copied from the selected source element(s) to the target element(s) by selecting **Element | Copy Data**.

1.3.1. Engine

1. **GENERAL**

   Click on the **General** sub-group folder and enter the following data:

   Engine Speed: 5000 rpm

   Cycle Type: 4 stroke

   BMEP Control: Activate

2. **CYLINDER / RPE-ROTOR SETUP**

   Click on the **Cylinder / RPE-Rotor Setup** sub-group folder to open the following window and enter the values shown.

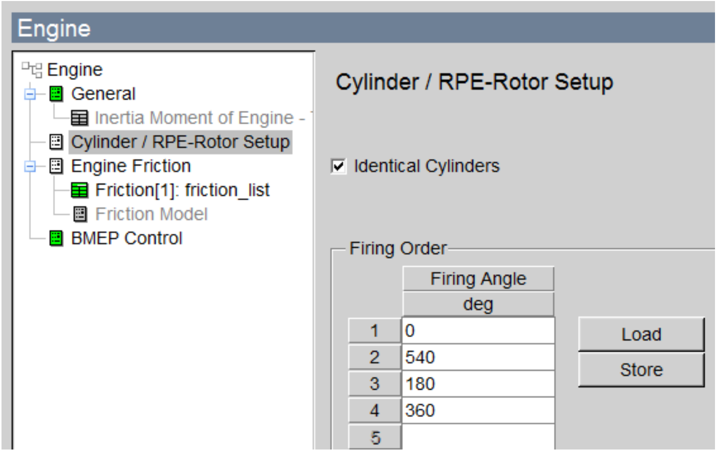


Figure 1-5: Engine – Cylinder / RPE-Rotor Setup Window

3. **ENGINE FRICTION**

In the **Engine Friction** sub-group folder, **Table** is selected as default. Click on the **Engine Friction[1]: friction_list** sub-group folder and enter the following data:

BMEP: 10 bar

Engine Speed (X) rpm	FMEP (Y) bar
1000	0.6
6000	2.3

Click **OK**.

4. **BMEP CONTROL**

Click on **BMEP Control** sub-group folder to open the following window. Enter the data as shown with **Flow Coefficients** as the controlled value and **Restriction 3** as the controlled element.

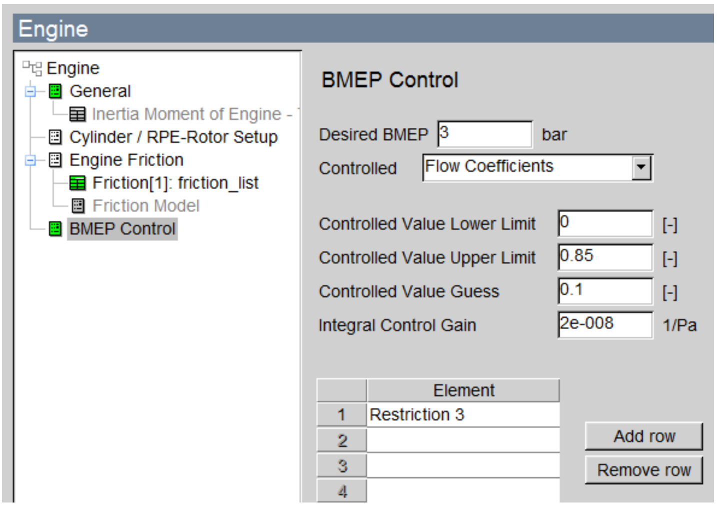


Figure 1-6: Engine – BMEP Control Window

Click **OK**.

1.3.2. Cylinder

Only the specifications for **Cylinder 1** have to be specified. All cylinders will have identical specifications. The direction of positive piston pin offset is defined as the direction of the rotation of the crankshaft at Top Dead Center (TDC).

The data for the cylinder is listed below. Click on the cylinder number to access the input fields.

1. **GENERAL**

   Click on the **General** sub-group folder and enter the following data:

Bore:	86 mm
Stroke:	86 mm
Compression Ratio:	10.5
Con-rod Length:	143.5 mm
Piston Pin Offset:	0 mm
Effective Blow by Gap:	0 mm
Mean Crankcase Press:	1 bar
Scavenge Model:	Perfect Mixing

2. **INITIALIZATION**

   Click on the **Initialization** sub-group folder and enter the following data:

   Initial Conditions at EO (Exhaust Valve Opening)

Pressure:	5 bar
Temperature:	726.85 degC

   Initial Gas Composition

Ratio Type:	A/F Ratio
Ratio Value:	14.3
Fuel Vapour:	0
Combustion Products:	1

3. **COMBUSTION**

   Click on the **Combustion** sub-group folder and select **Vibe** from the pull down menu for Heat Release.

   Click on the **Vibe** sub-group folder and enter the following data:

Start of Combustion:	-5 deg
Combustion Duration:	47 deg
Shaping Parameter m	1.6
Parameter a	6.9

   The following window displays the graphs of the Heat Release Characteristics.

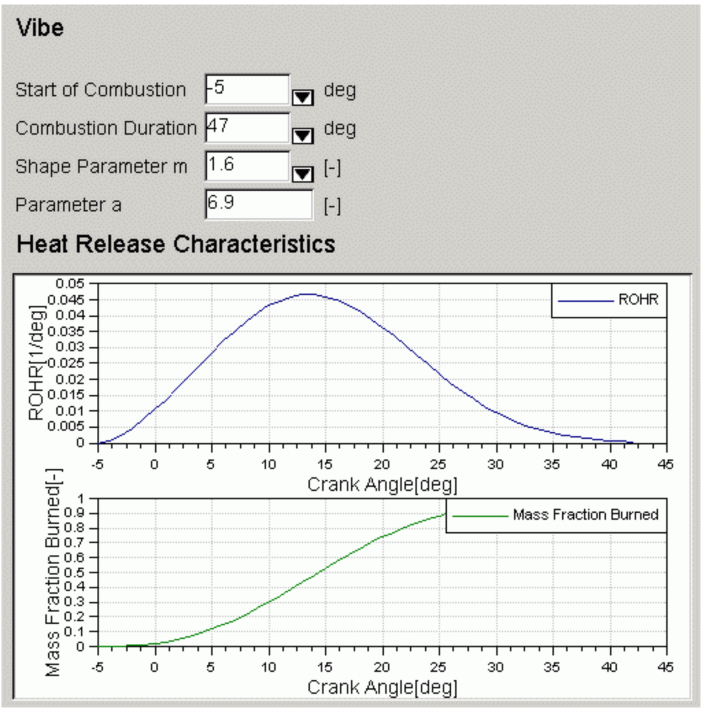


Figure 1-7: Vibe Window

4. **HEAT TRANSFER**

Click on the **Heat Transfer** sub-group folder and enter the following data:

Cylinder:	Woschni 1978	
Ports:	Zapf	
Piston:		
Surface Area:	5809 mm^2	
Wall Temperature:	226.85 degC	
Piston Calibration Factor:	1	
Cylinder Head:		
Surface Area:	7550 m^2	
Wall Temperature:	256.85 degC	
Head Calibration Factor:	1	
Liner:		
Surface Area:	270 mm^2	(Piston at TDC)
Wall Temperature:	161.85 degC	(Piston at TDC)
Wall Temperature:	151.85 degC	(Piston at BDC)
Liner Calibration Factor:	1	
Combustion System	DI	
Incylinder Swirl Ratio:	0	

5. VALVE PORT SPECIFICATIONS

For each pipe attached to a cylinder, the user has to specify whether this port is controlled by a valve or by the piston (piston control is only feasible for 2-stroke engines). If the cylinder features a combustion chamber, the pipe may be also declared to be attached to the chamber. In this case, the port may be either controlled by a valve or with the standard definition of flow coefficients.

If the heat transfer in the intake and exhaust ports has to be considered, the specification of the port surface area and of the mean port wall temperature is required (valve controlled port only).

For the calculation of the energy balance of the port wall similar data as for the combustion chamber walls (i.e. the average thickness, the heat capacity and the conductivity of the material) are required (only if wall temperature is set to variable).

For valve controlled ports, the inner valve seat diameter is required for the calculation of the port wall heat transfer coefficient, as well as for the conversion of normalized valve lift to effective valve lift.

Click on the **Valve Port Specification** sub-group folder and enter the following data for each cylinder:

Controlled by		**Port**	
Pipe	**Control**	**Surface Area mm^2**	**Wall Temp degC**
14	Valve	0	86.85
18	Valve	8300	266.85

In this example the intake pipe for cylinder 2 is 15 and the exhaust pipe is 19. The intake pipe for cylinder 3 is 16 and the exhaust pipe is 20 and the intake pipe for cylinder 4 is 17 and the exhaust pipe is 21.

Expand the **VPS [1]: Pipe 14 Intake** sub-group folder, click on **Valve Controlled** and enter the following data:

Inner Valve Seat (=Reference) Diameter	43.84 mm
Valve Clearance	0 mm
Scaling Factor for Eff. Flow Area	1.712

Expand the **Valve Controlled** folder, click on **Lift Curve** and enter the following data:

Specification		**Manipulation**	
Valve Opening	340 deg	**Valve Opening**	340 deg
Cam Length	270 deg	**Cam Length**	270 deg
Increment	10 deg		

Load the Intake Lift Curve from the installation:
`examples\BOOST\v2018 \spark_ignited\boost\data_ottocalc\`
`IV_lift_curve.dat`

Click on **Flow Coefficient** and enter the following data:

Pressure Ratio	1

Effective Valve Lift Active

Load the Intake Flow Coefficient curve from:

`…\data_ottocalc\IV_flow_coeff_curve.dat`

Expand the **VPS [2]: Pipe 18 Exhaust** sub-group folder, click on **Valve Controlled** and enter the following data:

Inner Valve Seat (=Reference) Diameter	36.77 mm
Valve Clearance	0 mm
Scaling Factor for Eff. Flow Area	1.242

Expand the **Valve Controlled** folder, click on **Lift Curve** and enter the following data:

Specification		**Manipulation**	
Valve Opening	130 deg	**Valve Opening**	130 deg
Cam Length	260 deg	**Cam Length**	260 deg
Increment	10 deg		

Load the Exhaust Lift Curve from the installation:
`…\data_ottocalc\EV_lift_curve.dat`

Click on **Flow Coefficient** and enter the following data:

Pressure Ratio	1
Effective Valve Lift	Active

Load the Intake Flow Coefficient curve from:

`…\data_ottocalc\IV_flow_coeff_curve.dat`

The following two options are available for modeling a multi-valve engine:

- A pipe is connected to each valve (Figure 1-8, left side):

  In the sketch, the branched part of the intake and exhaust port is modeled by two pipes and a junction. For this junction, the refined model should be used exclusively, as the constant pressure model causes to high pressure losses. This modeling is required only if the two valves feature different valve timings, the geometry of the runner attached to each valve is different or a valve deactivation systems is used.
- All intake and all exhaust valves are modeled by one pipe attachment (Figure 1-8, right side):

  The number of valves is taken into account by specifying the flow coefficients and scaling factor in such a way that the total effective flow area of all considered valves is obtained. This modeling is preferred as it requires fewer elements and is therefore less complicated and more efficient.

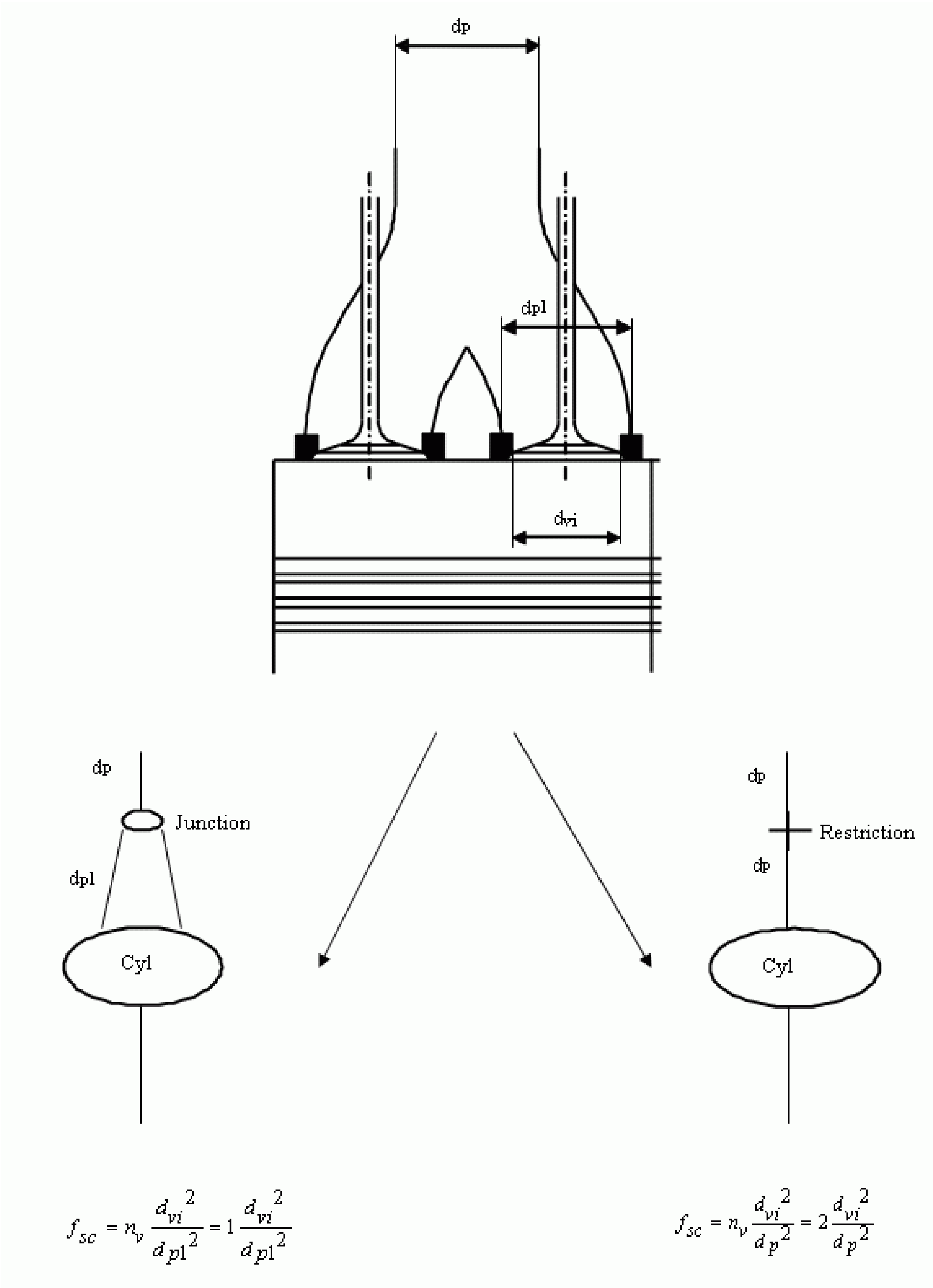


Figure 1-8: Modeling multi-valve engines

The ports will be modeled with one pipe each. In this case, the inner valve seat diameter of the intake valve is 31 mm. This gives a scaling factor of:

$$f_{sc} = \frac{n_v \cdot d_{vi}^2}{d_{pipe9}^2} = \frac{2 \cdot 31^2}{33.5^2} = 1.7115$$

Note that as the Flow Coefficients are specified just for one pressure ratio, they will be used irrespective of the actual pressure ratio across the valves.

1.3.3. Air Cleaner

The performance characteristics at the design point have to be specified in addition to the geometrical data. **BOOST** automatically creates a more refined calculation model of a plenum-pipe-plenum type for the air cleaner. This is used to model the gas dynamic performance of the air cleaner as well as the pressure drop over the air cleaner depending on the actual flow conditions.

The data for the air cleaner is listed below. Click on the air cleaner number to access the input fields.

1. **GENERAL**

   Click on the **General** sub-group folder and enter the following data:

   Geometrical Properties

Total Air Cleaner Volume:	8.7 (l)
Inlet Collector Volume:	3.0 (l)
Outlet Collector Volume:	4.3 (l)
Length of Filter Element:	300 mm

   Friction Specification Target Pressure Drop

   Target Pressure Drop

Mass Flow	0.13 kg/s
Target Pressure Drop	0.008 bar
Inlet Pressure	1 bar
Inlet Air Temperature	19.85 degC

   The length of the filter element is also used to model the time a pressure wave needs to travel through the cleaner.

2. **FLOW COEFFICIENTS**

   Click on the **Flow Coefficients** sub-group folder and enter the following data:

Pipe 3 Inflow	1	**Pipe 3 Outflow**	1
Pipe 4 Inflow	1	**Pipe 4 Outflow**	1

   The air cleaner performance is specified by means of a reference mass flow, the target pressure drop (defined as the static pressure difference at the inlet and the outlet pipe attachment) at the reference mass flow and the inlet air conditions (temperature and pressure), refer to the following figure.

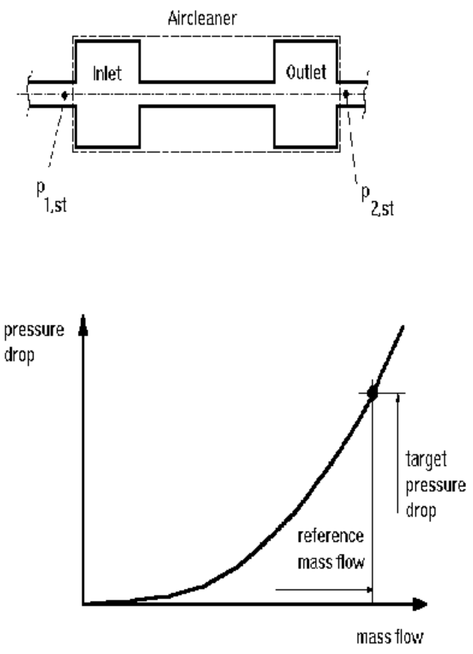


Figure 1-9: Steady State Air Cleaner Performance

On the basis of this information, the wall friction loss of the model is adjusted by the program.

1.3.4. Catalyst

NOTE

The catalyst model is a purely gas dynamic model and does not include chemical reactions.

The data for the catalyst is listed below. Click on the catalyst number to access the input fields.

1. **GENERAL**

Click on the **General** sub-group folder and enter the following data:

Chemical Reactions	Deactivate
Monolith Volume:	3.2 (l)
Length of Monolith:	300 mm
Inlet Collector Volume:	0.15 (l)
Outlet Collector Volume:	0.15 (l)

2. **TYPE SPECIFICATION**

Click on the **Type Specification** sub-group folder and enter the following data:

Catalyst Type Specification

General Catalyst	Activate

General Catalyst

Open Frontal Area (OFA)	1
Hydraulic Unit	Diameter
Hydraulic Diameter	116.5385 mm
Geometrical Surface Area (GSA)	34.32421 1/m (calculated)

3. **FRICTION**

Click on the **Friction** sub-group folder and enter the following data:

Friction Specification

Target Pressure Drop	Activate

Target Pressure Drop

Inlet Massflow	0.13 kg/s
Inlet Temperature	826.85 degC
Inlet Pressure	1.4 bar
Target Pressure Drop	0.22 bar

4. **FLOW COEFFICIENTS**

Click on the **Flow Coefficients** sub-group folder and enter the following data:

Pipe 31 Inflow	1	**Pipe 31 Outflow**	1
Pipe 32 Inflow	1	**Pipe 32 Outflow**	1

1.3.5. Injector

To consider the particular pressure losses resulting from multi-dimensional flow phenomena which cannot be predicted by the program, **BOOST** requires the specification of flow coefficients at the fuel injector. The flow coefficients are defined as the ratio between the actual mass flow and a loss-free isentropic mass flow for the same stagnation pressure and the same pressure ratio.

The fuel supply is specified by the A/F ratio.

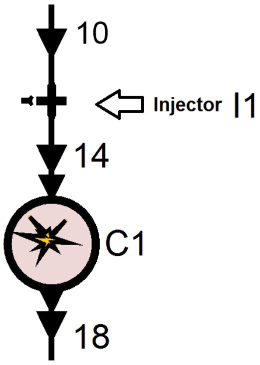
If the **Carburettor model** is used, the instantaneous mass flow at the carburetor position is used together with the specified A/F ratio to calculate the amount of fuel supplied. Due to oscillating flow at the carburetor location a considerable enrichment of the mixture may occur.

NOTE

It is necessary to check the actual A/F ratio in the cylinder and to correct the A/F ratio at the carburetor if the values are different to those desired.

For the **Injection Nozzle (Continuous Injection) model**, a measuring point at the position of the air flow meter must be specified. In this case the fueling is calculated from the mass flow at the air flow meter position and the specified A/F ratio. As the air flow meter usually serves more than one injector, the percentage of the total air flow served by each injector must be specified.

As shown in the following figure, the arrow of the injector is at the left side for a flow condition from top to down before the cylinder:



The data for the four injectors is the same and listed below. Data can be copied from one injector to others by selecting **Element | Copy Data**. Click on the injector number to access the input fields.

1. **GENERAL**

   Click on the **General** sub-group folder and select **Continuous** as the injection method.

2. **MASS FLOW**

   Click on the **Mass Flow** sub-group folder, select **Ratio Control** and enter the following data for each injector:

Air Fuel Ratio:	13.34	
Injector Model:	Injection Nozzle (Continuous Injection)	
Air Flow taken from Measuring Point:	Measuring Point 2	
The Inject Covers	25%	of the Total Air Flow

3. **FLOW COEFFICIENTS**

   Click on the **Flow Coefficients** sub-group folder and enter the following data.

Injector 1	**from Pipe 10 to Pipe 14**	0.95
	from Pipe 14 to Pipe 10	0.95
Injector 2	**from Pipe 11 to Pipe 15**	0.95
	from Pipe 15 to Pipe 11	0.95

Injector 3	**from Pipe 12 to Pipe 16**	0.95
	from Pipe 16 to Pipe 12	0.95
Injector 4	**from Pipe 13 to Pipe 17**	0.95
	from Pipe 17 to Pipe 13	0.95

1.3.6. System Boundary

The system boundary provides the connection of the calculation model to a user-definable ambient. The ambient conditions (pressure, temperature, air/fuel ratio, fuel vapor, and combustion products) have to be specified either as constant values, or as functions of time or crank angle.

If internal mixture preparation is considered, the input of fuel vapor and combustion products is disabled. In this case the A/F ratio represents the A/F ratio of the mixture of air and combustion products in the ambient, and no unburned fuel in the ambient is allowed.

If external mixture preparation is considered, the A/F ratio represents the A/F ratio of the combustion gases in the ambient. In addition, the mass fractions of the combustion products and of the fuel vapor have to be specified.

The data for each system boundary is listed below. Data can be copied from one system boundary to others by selecting **Element | Copy Data**. Click on the relevant SB number to access the input fields.

1. **GENERAL**

   Click on the **General** sub-group folder and select **Standard** for **Boundary Type**.

2. **BOUNDARY CONDITIONS**

   Click on the **Boundary Conditions** sub-group folder and enter the following data:

   Select **Local Boundary Conditions.** Select **Set 1** for **Preference** (defined in section 1.2. – Initialization).

	Pressure (bar)	**Gas Temp (degC)**	**Fuel Vapour**	**Combustion Products**	**Ratio Type**	**Ratio Value**
SB 1	0.995	30.85	0	0	A/F Ratio	10000
SB 2	0.995	676.85	0	1	A/F Ratio	14.3

3. **FLOW COEFFICIENTS**

   Click on the **Flow Coefficients** sub-group folder and enter the following data:

SB 1	**Pipe 1 Inflow**	0.95	**Pipe 1 Outflow**	0.95
SB 2	**Pipe 34 Inflow**	1	**Pipe 34 Outflow**	1

1.3.7. Plenum

The data for the plenums is listed below. Data can be copied from one plenum to others by selecting **Element | Copy Data**. Click on the relevant plenum number to access the input fields.

1. **GENERAL**

   Click on the **General** sub-group folder and enter the relevant volume for each plenum. Click on the **Initialization** sub-group folder and select **Initialization Preferences** for each plenum. Select the relevant set from the **Preference** pull-down menu.

	Volume (l)	**Global Initialization**
Plenum 1	2.6	Set 1
Plenum 2	4	Set 1
Plenum 3	6	Set 3
Plenum 4	6	Set 3

2. **FLOW COEFFICIENTS**

   Click on the **Flow Coefficients** sub-group folder and enter the following data:

Plenum 1	**Pipe 2 Inflow**	0.9	**Pipe 2 Outflow**	0.9
Plenum 2	**Pipe 6 Inflow**	0.95	**Pipe 6 Outflow**	0.95
	Pipe 7 Inflow	0.95	**Pipe 7 Outflow**	0.95
Plenum 3	**Pipe 32 Inflow**	0.6	**Pipe 32 Outflow**	0.6
	Pipe 33 Inflow	0.65	**Pipe 33 Outflow**	0.65
Plenum 4	**Pipe 33 Inflow**	0.9	**Pipe 33 Outflow**	0.9
	Pipe 34 Inflow	0.9	**Pipe 34 Outflow**	0.9

The reduced flow coefficients at the attached Pipe 4 accounts for the flow losses of the opened throttle. The throttle is not explicitly modeled in this example.

Plenums 3 and 4 model mufflers. The flow coefficients were reduced to account for the pressure losses caused by the inside of the mufflers.

1.3.8. Junctions

The **Refined Model** requires flow coefficients for each flow path in each possible flow pattern. For the three-way junction, this adds up to two times six flow coefficients. The following figure shows the qualitative trend of these flow coefficient versus the ratio of the mass flow in a single branch to the mass flow in the common branch for a joining flow pattern.

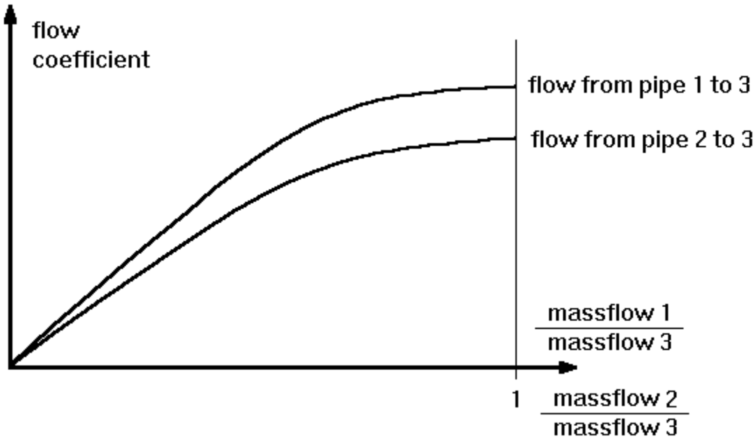


Figure 1-10: Flow Coefficients of a Junction

The actual values depend on the geometry of the junction, i.e. the area ratio and the angle between the pipes. **BOOST** interpolates suitable flow coefficients for the considered junction from a database (`RVALF.CAT`) delivered with the **BOOST** code. The database contains the flow coefficients of six junctions, covering a wide range of area ratios and angles. The data was obtained by measurements on a steady state flow test rig. The file `RVALF.CAT` is a formatted ASCII file. The user may add measured flow coefficients for special junctions or for an extension of the catalogue.

The data for the junctions is listed below. Data can be copied from one junction to others by selecting **Element|Copy Data**. Click on the relevant junction number to access the input fields.

1. **GENERAL**

Click on the **General** sub-group folder and select the relevant Junction type. Then select the relevant sub-group in the tree and enter the following data:

<table>
<tr><td>Junction 1</td><td>Refined Model</td><td colspan="3">Angle between Pipes 1 and 2
Angle between Pipes 2 and 3
Angle between Pipes 3 and 1</td><td>135
45
180</td></tr>
<tr><td>Junction 2</td><td>Constant Pressure</td><td>Pipe 8 Inflow
Pipe 9 Inflow
Pipe 10 Inflow
Pipe 13 Inflow</td><td>1
1
1
1</td><td>Pipe 8 Outflow
Pipe 9 Outflow
Pipe 10 Outflow
Pipe 13 Outflow</td><td>1
1
1
1</td></tr>
<tr><td>Junction 3</td><td>Refined Model</td><td colspan="3">Angle between Pipes 9 and 11
Angle between Pipes 11 and 12
Angle between Pipes 12 and 9</td><td>90
180
90</td></tr>
<tr><td>Junction 4</td><td>Refined Model</td><td colspan="3">Angle between Pipes 22 and 23
Angle between Pipes 23 and 26
Angle between Pipes 26 and 22</td><td>30
165
165</td></tr>
<tr><td>Junction 5</td><td>Refined Model</td><td colspan="3">Angle between Pipes 24 and 25
Angle between Pipes 25 and 27
Angle between Pipes 27 and 24</td><td>30
165
165</td></tr>
<tr><td>Junction 6</td><td>Refined Model</td><td colspan="3">Angle between Pipes 28 and 29
Angle between Pipes 29 and 30
Angle between Pipes 30 and 28</td><td>30
165
165</td></tr>
</table>

A plane junction is assumed. Therefore, the sum of all three angles needs to be 360 degrees.

1.3.9. Restrictions

The flow coefficients are defined as the ratio between the actual mass flow and the loss-free isentropic mass flow for the same stagnation pressure and the same pressure ratio.

NOTE

In **BOOST** the flow coefficients of restrictions are always related to the minimum attached pipe cross-section.

The data for the restrictions is listed in the following table. Data can be copied from one restriction to others by selecting **Element** | **Copy Data**. Click on the relevant restriction number to access the input fields.

FLOW COEFFICIENTS

Click on the **Flow Coefficients** sub-group folder and enter the following data:

Restriction 1	**from Pipe 4 to Pipe 5**	1
	from Pipe 5 to Pipe 4	1
Restriction 2	**from Pipe 5 to Pipe 6**	1
	from Pipe 6 to Pipe 5	1
Restriction 3	**from Pipe 7 to Pipe 8**	0.85
	from Pipe 8 to Pipe 7	0.85
Restriction 4	**from Pipe 18 to Pipe 22**	0.8
	from Pipe 22 to Pipe 18	0.8
Restriction 5	**from Pipe 19 to Pipe 24**	0.8
	from Pipe 24 to Pipe 19	0.8
Restriction 6	**from Pipe 20 to Pipe 25**	0.8
	from Pipe 25 to Pipe 20	0.8
Restriction 7	**from Pipe 21 to Pipe 23**	0.8
	from Pipe 23 to Pipe 21	0.8
Restriction 8	**from Pipe 26 to Pipe 28**	0.9
	from Pipe 28 to Pipe 26	0.9
Restriction 9	**from Pipe 27 to Pipe 29**	0.9
	from Pipe 29 to Pipe 27	0.9
Restriction 10	**from Pipe 30 to Pipe 31**	0.9
	from Pipe 31 to Pipe 30	0.9

1.3.10. Pipes

One dimensional flow is calculated in the pipes by solving the appropriate equations. This means that the pipe is the only element where the time lag caused by the propagation of pressure waves, or the flow itself, is considered.

The heat transfer coefficient for the calculation of the heat flux from or to the pipe walls is calculated from the Reynolds' analogy. The heat transfer factor allows the user to increase or to reduce the heat transfer as the calculated heat transfer coefficient is multiplied by this factor.

Modeling of the ports deserves special attention. The flow coefficients are measured in an arrangement like that shown in the following figure.

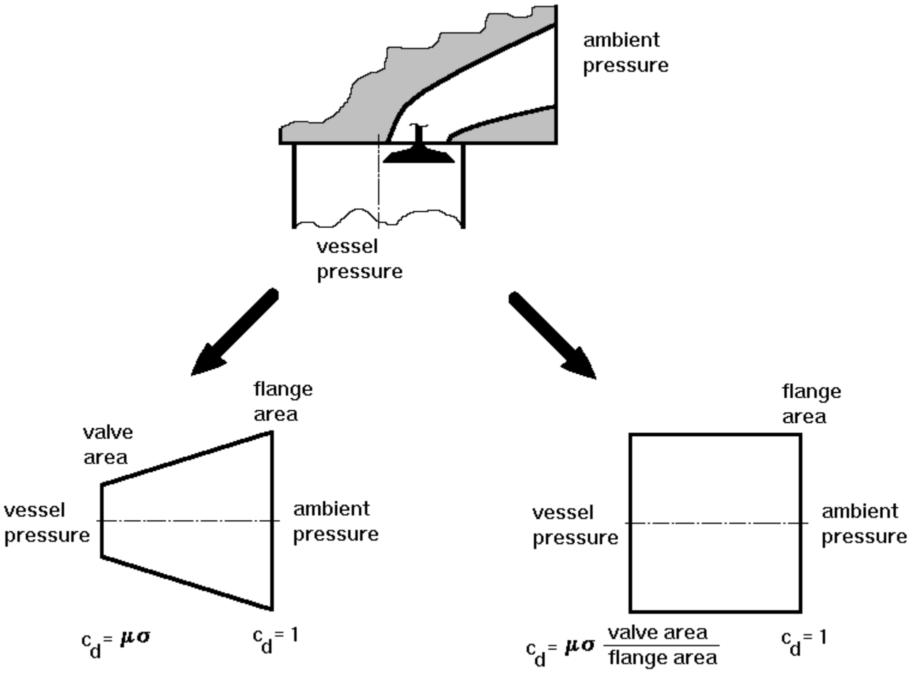


Figure 1-11: Flow Coefficient Measurement

The measured mass flow rate is related to the isentropic mass flow rate calculated with the valve area and the pressure difference across the port. The obvious model as shown in the same Figure on the bottom left side would produce mass flow rates which are too high (too low in case of a nozzle shaped exhaust port), because the diffuser modeled causes a pressure recovery increasing the pressure difference at the entry of the pipe modeling the port. The mass flow rate is calculated with the increased pressure difference and the valve area, and is therefore greater than the measured one. This problem can be overcome either by a correction of the flow coefficients or by switching to a model as shown in Figure 1-11 on the bottom right side. Due to the modeling of the pipe as a straight diameter pipe with flange area, there is no pressure recovery. However, the flow coefficients need to be corrected by the ratio of the different areas. This can be done easily by the scaling factor. For this model, the Constant Diameter Model is used.

The data for each pipe is listed in the following tables. Data can be copied from one pipe to others by selecting **Element**|**Copy Data**. Click on the relevant pipe number to access the input fields.

1. Click on the **General** sub-group folder and enter the following data for each pipe. The default **Bending Radius** (100000 mm) is used.

2. In the **Initialization** sub-group, select the required **Global** set from the **Preference** pull-down menu.

	Pipe Length (mm)	Diameter (mm)	Friction Coeff.	Heat Transfer Factor	Wall Temp (degC)	Global Initial.
Pipe 1	110	TABLE	0.001	1	30.85	Set 1
Pipe 2	140	45	0.019	1	30.85	Set 1
Pipe 3	220	TABLE	0.001	1	30.85	Set 1
Pipe 4	220	TABLE	0.01	1	30.85	Set 1
Pipe 5	60	60	0.01	1	30.85	Set 1
Pipe 6	60	100	0.01	1	30.85	Set 1
Pipe 7	40	70	0.034	1	30.85	Set 1
Pipe 8	105	TABLE	0.034	1	30.85	Set 1
Pipe 9	80	TABLE	0.034	1	19.85	Set 1
Pipes 10 - 13	320	TABLE	0.036	1	36.85	Set 1
Pipes 14 - 17	100	33.5	0.04	1	66.85	Set 2
Pipe 18	80	32	0.04	1	576.85	Set 3
Pipes 19 - 21	80	32	0.04	1	576.85	Set 3
Pipe 22	305	32	TABLE	1	576.85	Set 3
Pipe 23	285	32	TABLE	1	576.85	Set 3
Pipe 24	300	32	TABLE	1	576.85	Set 3
Pipe 25	270	32	TABLE	1	576.85	Set 3
Pipe 26	50	34	0.023	1	576.85	Set 3
Pipe 27	50	35	0.023	1	576.85	Set 3
Pipe 28	360	37	0.022	1	476.85	Set 3
Pipe 29	290	37	0.022	1	476.85	Set 3
Pipe 30	50	44	0.021	1	576.85	Set 3
Pipe 31	970	TABLE	TABLE	1	326.85	Set 3
Pipe 32	860	46	0.021	1	476.85	Set 3
Pipe 33	970	46	0.021	1	426.85	Set 3
Pipe 34	330	46	0.021	1	376.85	Set 3

Enter the following data for the relevant Table.

	Diameter		Bending Radius		Friction Coefficient	
	Location X (mm)	**Diameter Y (mm)**	**Location X (mm)**	**Bending Radius Y (mm)**	**Location X (mm)**	**Friction Coeff Y**
Pipe 1	0	55				
	110	44				
Pipe 3	0	44				
	220	80				
Pipe 4	0	70				
	110	60				
	220	60				
Pipe 8	0	75				
	52.5	75				
	105	65				
Pipe 9	0	65				
	80	55				
Pipes 10 - 13	0	45	0	0		
	70	42.7	105	120		
	115	41.3	210	60		
	170	36.8	320	10000		
	225	33.5				
	265	33.4				
	320	33.4				
Pipe 22					0	0.04
					102.5	0.04
					213.75	0.023
					305	0.023
Pipe 23					0	0.04
					102.5	0.04
					213.75	0.023
					285	0.023
Pipe 24					0	0.04
					110	0.04
					205	0.023
					300	0.023
Pipe 25					0	0.04
					115	0.04
					182.5	0.023
					270	0.023
Pipe 31	0	46	0	0	0	0.019
	220	46	220	10000	220	0.06
	400	45	570	100	570	0.06
	570	46	970	10000	970	0.019
	970	46				

1.3.11. Measuring Point

Measuring points allow the user to access flow data and gas conditions over crank angle at a certain location in a pipe. The location of the measuring point has to be specified as its distance from the upstream pipe end. The user may select the output for a measuring point. By selecting **Standard Output**, pressure, flow velocity, temperature, Mach number and mass flow rates are available. If **Extended Output** is selected, the following data are available in addition: stagnation pressure, stagnation temperature, enthalpy flow, fuel concentration, combustion products concentration, fuel flow, and combustion products flow.

The data for the measuring points is listed in the following table. Data can be copied from one measuring point to others by selecting **Element** | **Copy Data**. Click on the relevant measuring point number to access the input fields.

1. **GENERAL**

   Click on the **General** sub-group folder and enter the following data:

	Location of Measuring Point from Upstream Pipe End (mm)	Output Extent
Measuring Point 1	170	Standard
Measuring Point 2	35	Standard
Measuring Point 3	200	Standard
Measuring Point 4	60	Standard
Measuring Point 5	40	Standard
Measuring Point 6	25	Standard
Measuring Point 7	40	Standard
Measuring Point 8	275	Standard
Measuring Point 9	268	Standard
Measuring Point 10	50	Standard
Measuring Point 11	50	Standard
Measuring Point 12	180	Standard
Measuring Point 13	200	Standard
Measuring Point 14	260	Standard
Measuring Point 15	50	Standard
Measuring Point 16	900	Standard
Measuring Point 17	50	Standard
Measuring Point 18	305	Standard

1.3.12. Reference Point for Volumetric Efficiency

The BOOST pre-processor allows a plenum or a measuring point to be specified as reference location for the calculation of the air delivery ratio and the volumetric efficiency related to intake manifold conditions.

Select **Simulation** | **Volumetric Efficiency** to open the following window. In this example select **Measuring Point 2** as the reference element.

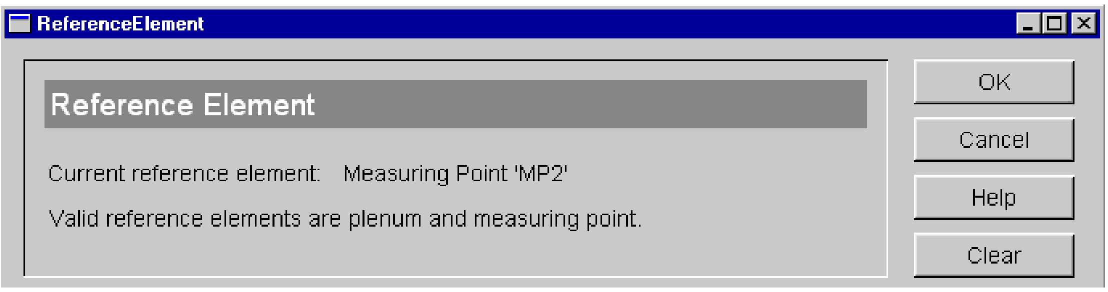


Figure 1-12: Reference Point for Volumetric Efficiency

NOTE

Save the model before starting calculation.

1.4. Run Simulation

Select **Simulation** | **Run** and then select the required case and tasks to be run. Select **Run** to start the simulation.

A window opens which provides an overview of the status of the simulation. Double click on the row of the desired simulation to view a more detailed information of the simulation run produced by the simulation kernel. Once the job is complete select **OK** to exit this window and then **Close** to exit the simulation status window.

1.5. Post-processing

Select **Simulation** | **Show Messages** | **Cycle Simulation** to check for convergence warnings and relevant information.

Select **Simulation** | **Show Summary** | **Cycle Simulation** to check summary information about the simulation run, e.g. overall engine performance.

Select **Simulation** | **Show Results** to open **IMPRESS Chart**. Select the **Results** tab to display the tree and select the **Traces** folder with the right mouse button. Select **Model view** from the submenu to display the model below the tree.

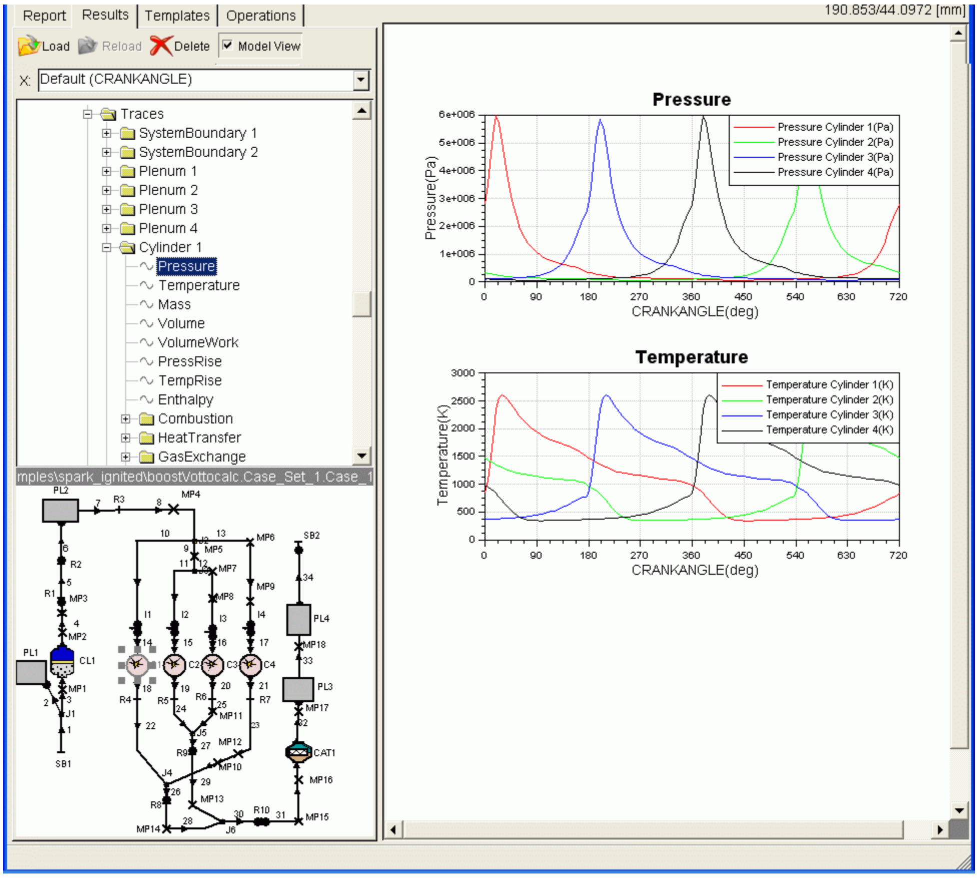


Figure 1-13: Results Window

Click on the **Layer** and then select **Pressure** from Cylinder 1. In the model tree double click on cylinder 2, 3 and 4 and add the relevant curves. In this example temperature is displayed using the same procedure.

1.6. General Species Transport

1.6.1. ottocalc_species.bwf

This example is setup identical to the `ottocalc.bwf` except that the general species transport option is used.

1. In the **Simulation | Control – Cycle Simulation** window, select **General** for **Species Transport**. Then select **General Species Setup** sub-group folder in the tree to access the following window:

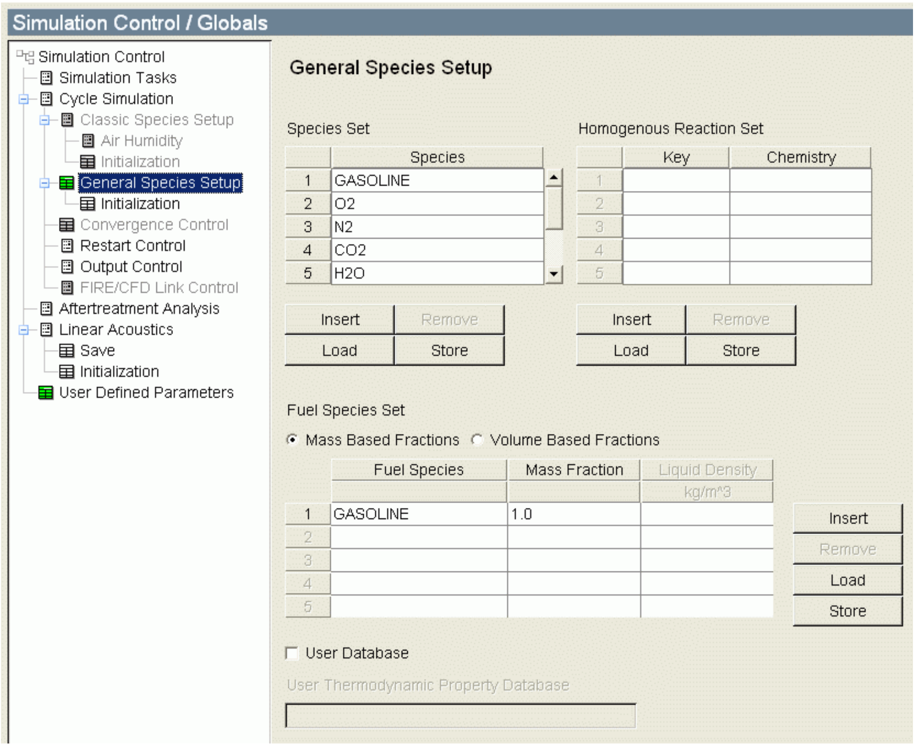


Figure 1-14: Simulation Control – General Species Setup Window

Enter the following data:

Species Set:	GASOLINE, O2, N2, CO2, H2O, CO, H2, O, NO
Fuel Species:	GASOLINE
Mass Fraction:	1.0

2. In the **Cylinder - Combustion** window, ensure the following options under **General Species** are disabled:

   Single Zone Chemistry

   Gas Exchange Phase Chemistry

   Solver absolute Tolerance

   Solver relative Tolerance

3. In the **Injector – Species Options** window, **Fuel** is activated and **Consider Heat of Evaporation** is deactivated as default.

1.6.2. ottocalc_species_2zone.bwf

This example is setup identical to the `ottocalc.bwf` except that the general species transport option is used and the `ottocalc.bwf` is converted to a two-zone model taking into account detailed pollutant formation chemistry in the burned zone.

1. In the **Simulation | Control – Cycle Simulation** window, select **General** for **Species Transport**. Then select **General Species Setup** sub-group folder in the tree to access the following window:

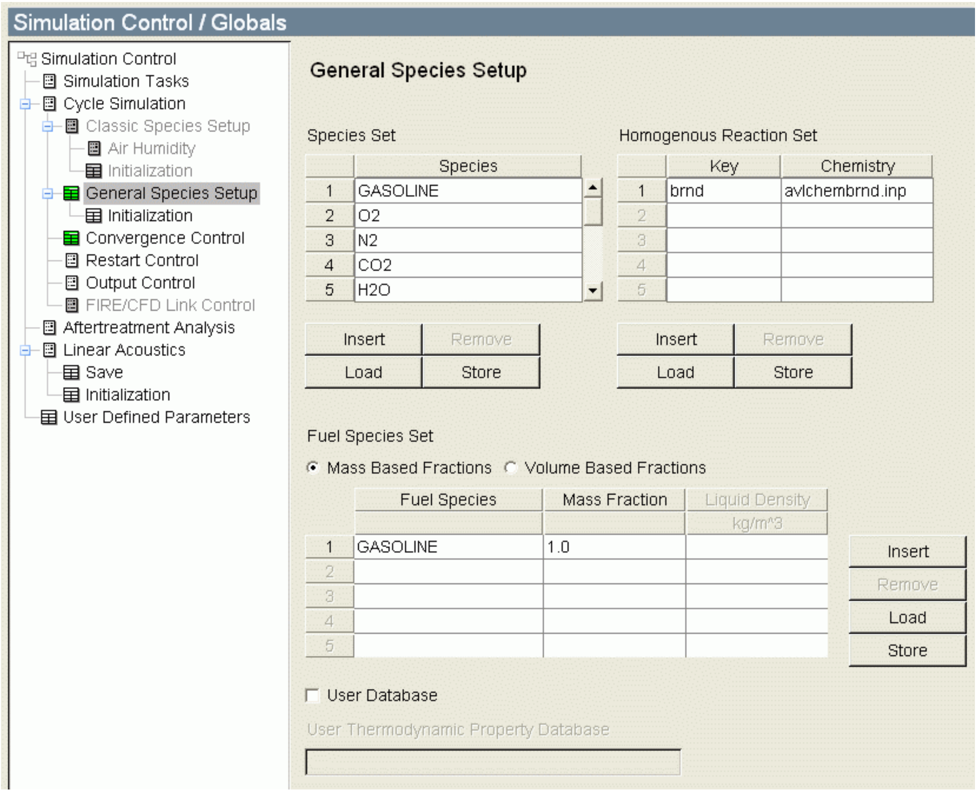


Figure 1-15: Simulation Control – General Species Setup Window

Enter the following data:

Species Set: GASOLINE, O2, N2, CO2, H2O, CO, H2, O, OH H, HO2, H2O2, N, NO, N2O

Homogenous Reaction Set:

Key brnd

Chemistry: avlchembrnd.inp

Fuel Species: GASOLINE

Mass Fraction: 1.0

2. In the **Cylinder - Combustion** window, select **Vibe 2-Zone** for **Heat Release.**

   Then enable/disable the following options under **General Species**:

   Single Zone Chemistry enable and select **brnd**

   Gas Exchange Phase Chemistry disabled

Two Zone Unburned Chemistry	disabled
Two Zone Burned Chemistry	enable and select **brnd**
Solver absolute Tolerance	disabled
Solver relative Tolerance	disabled

3. In the **Cylinder – Vibe 2-Zone** window, enter the following data:

Start of Combustion:	-5 deg
Combustion Duration:	47 deg
Shaping Parameter m	1.6
Parameter a	6.9

4. In the **Cylinder – Pollutants** window, keep the default data.
5. In the **Injector – Species Options** window, **Fuel** is activated and **Consider Heat of Evaporation** is deactivated as default.

1.7. Case Series Calculations

1.7.1. Continuous Variable Valve Timing (CVVT)

Using the `ottocalc.bwf` single case model created in Chapter 1, the user can create a case series calculation by extending the cylinder with continuous variable valve timing (CVVT). The `otto_cvvt.bwf` file is used for this example and is located in the `spark_ignited` folder.

Parameters are assigned to a set of cases so that a series of variants can be calculated at one time.
The user is recommended to refer to Chapter 2.7 of the BOOST Users Guide for more detailed information on the case series calculation.

1.7.1.1. Assign New Parameter

1. Open the **Cylinder** element properties window.
2. Under the **Valve Port Specification** folder, click on **VPS[1]: Pipe 14: Intake** and select **Valve Controlled**.
3. Under **Modification of Valve Lift Timing**, activate the check box for **Valve Opening Shift**.
4. Click on the **Valve Opening Shift** label (text to the left of the input box) with the right mouse button and select **Assign new parameter (global)** from the submenu.

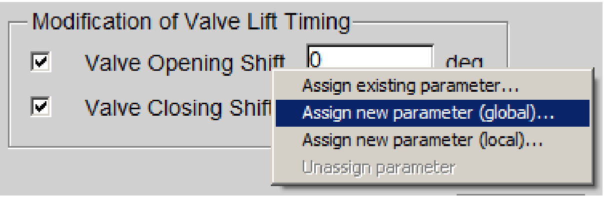


Figure 1-16: Assigning New Parameter

5. Enter `CVVT` for the name of the parameter in the dialog box. As all cylinders are identical, this parameter is the same for all.

6. Click on the **Valve Closing Shift** label with the right mouse button and select **Assign existing parameter (global)** from the submenu.

7. Select **CVVT** from the **Select Parameter** tree. Next the values for CVVT will be defined.

NOTE

Parameter names should not have any spaces.

8. Select **Model**|**Case Explorer** and **Case 1** is shown as the active case as it has a red circle.

9. Select  or **Group**|**Edit** to show the available parameters in the model. Select the `CVVT` parameter from the Unused Parameter list and click  to display it in the Used Parameters window. Select OK.

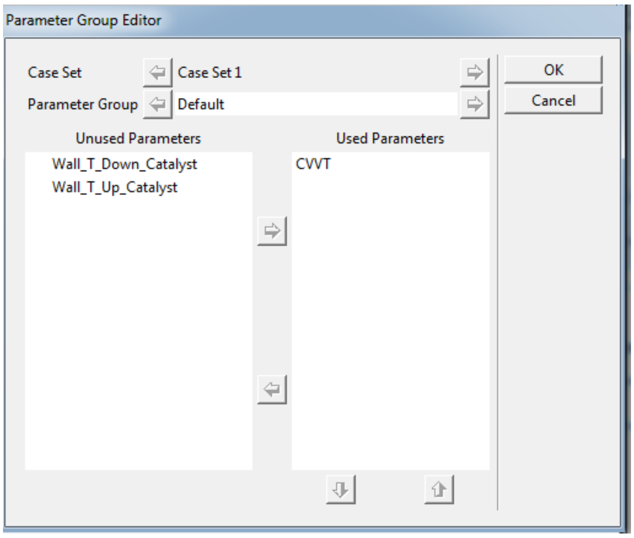


Figure 1-17: Parameter Group Editor Window

10. Select  or **Insert**|**Case** twice to add two cases.

11. Enter the values for each case as shown in the following figure.

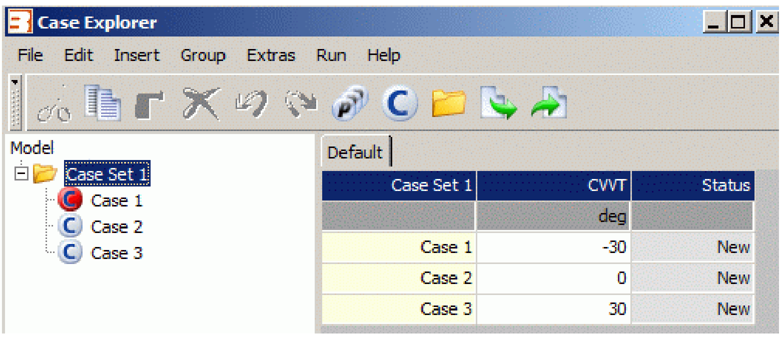


Figure 1-18: Case Explorer Window

1.7.1.2. Run Simulation

1. After saving the model, select **Simulation | Run** to open the following window.

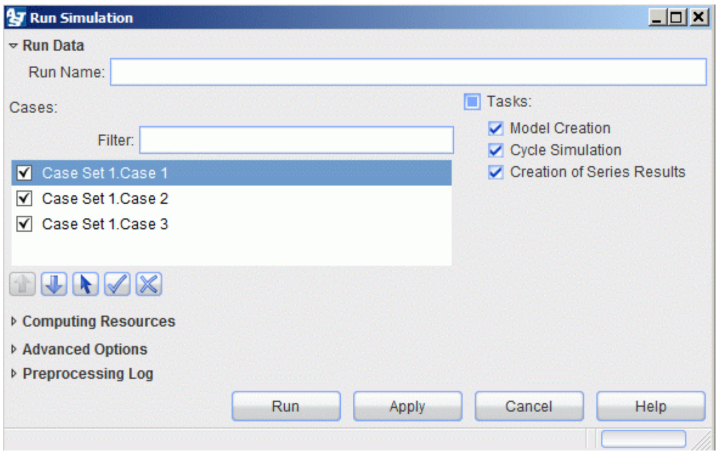


Figure 1-19: Run Simulation Window

2. Select all required cases and tasks. Then select **Run**. The progress of the simulation of each case can be followed in the Simulation Status and Task Information windows**.**

1.7.1.3. Post-processing

After successful completion of the simulation of each case, the series results can be examined in IMPRESS Chart.

1. Select **Simulation│Show Results.**

2. In IMPRESS Chart the series results can be plotted, e.g. volumetric efficiency versus intake valve shift, as shown in the following figure.

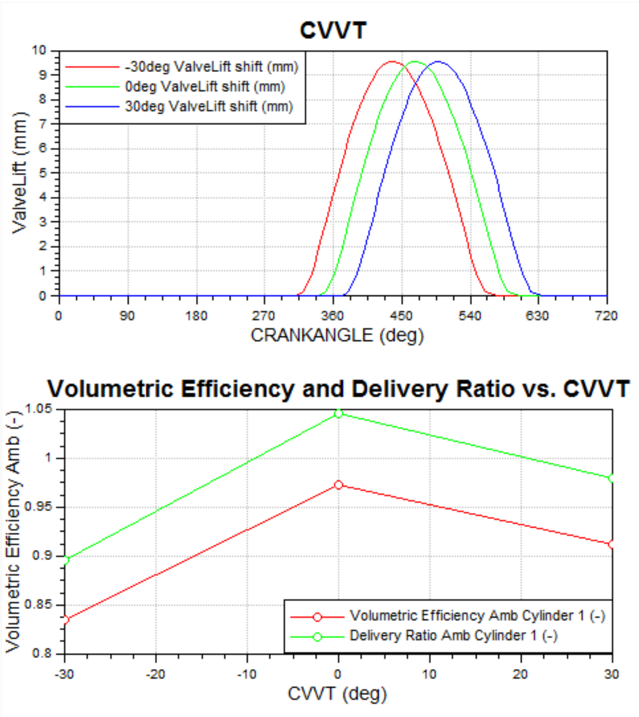


Figure 1-20: Intake Valve Shift and Effect of Intake Valve Shift on Volumetric Efficiency

1.7.2. Multiple Parameter Example - Ottoser.bwf

Using the `ottocalc.bwf` single case model created in Chapter 1, the user can create a case series calculation. The `ottoser.bwf` file is used for this example and is located in the `spark_ignited` folder.

Parameters are assigned to a set of cases so that a series of operating points or engine variants can be calculated at one time.

The user is recommended to refer to Chapter 2.7 of the BOOST Users Guide for more detailed information on the case series calculation.

1.7.2.1. Assign New Parameters

Firstly the parameters must be set for Case 1.

1. Open the **Engine** element properties window.
2. Click on the **Engine Speed** label (text to the left of the input box) with the right mouse button and select **Assign new parameter (global)** from the submenu.
3. Enter `Engine_Speed` for the name of the parameter in the dialog box.

NOTE

Parameter names should not have any spaces.

4. Repeat this procedure for each of the following parameters. The path locates the parameter from the relevant Properties window. For the cylinders the data should be input for cylinder number 1 (C1) and then later copied to the other cylinders (see below). As some parameters are used in more than one location, the first time a parameter is used select **Assign new parameter (global)** from the submenu. Select **Assign existing parameter** for subsequent use.

Parameter	Description	Path
AF_Ratio	Injected air/fuel ratio for each injector	Injector 1 (Injector / General / <Air Fuel Ratio>) Injector 2 (Injector / General / <Air Fuel Ratio>) Injector 3 (Injector / General / <Air Fuel Ratio>) Injector 4 (Injector / General / <Air Fuel Ratio>)
AF_Ratio	Initial air/fuel ratio in the cylinder	Cylinder / Initialization / <Ratio Value>
Start_of_Combustion	Start of combustion	Cylinder / Combustion / Vibe / <Start of Combustion>
Duration_of_Combustion	Duration of combustion	Cylinder / Combustion / Vibe / <Combustion Duration>
Vibe_Shape	Vibe shape parameter 'm'	Cylinder / Combustion / Vibe / <Shape parameter m>
Cylinder_Head_T	Cylinder head temperature	Cylinder / Heat Transfer / <Cylinder Head: Wall Temp.>

Liner_TDC_T	Liner temperature at TDC	Cylinder / Heat Transfer / <Liner: Wall Temp. (Piston at TDC)>
Liner_BDC_T	Liner temperature at BDC	Cylinder / Heat Transfer / <Liner: Wall Temp. (Piston at BDC)>
Piston_T	Piston temperature	Cylinder / Heat Transfer / <Piston: Wall Temperature>
Exhaust_Port_Wall_T	Exhaust port wall temperature	Cylinder / Valve Port Specification / <Port : Wall Temp : *Line for pipe 18*>
Exhaust_Wall_1_T	Exhaust wall temperature for first stage of exhaust manifold. (i.e. first pipes after exhaust port runners.)	Pipe 22 (Pipe / General / Wall Temperature) Pipe 23 (Pipe / General / Wall Temperature) Pipe 24 (Pipe / General / Wall Temperature) Pipe 25 (Pipe / General / Wall Temperature)
Exhaust_Wall_2_T	Exhaust wall temperature for second stage of exhaust manifold.	Pipe 26 (Pipe / General / Wall Temperature) Pipe 27 (Pipe / General / Wall Temperature)
Exhaust_Wall_3_T	Exhaust wall temperature for third stage of exhaust manifold.	Pipe 28 (Pipe / General / Wall Temperature) Pipe 29 (Pipe / General / Wall Temperature) Pipe 30 (Pipe / General / Wall Temperature) Pipe 31 (Pipe / General / Wall Temperature)

5. For pipes 32, 33 and 34 the wall temperature is defined to vary along the length of the pipe in this case series example. Firstly, define the wall temperature for the pipe as a Table and enter initial values for the wall temperature at each end of the pipe. These can be set to the constant value previously defined for the pipe. Then assign parameters to control these temperatures as shown in the following figure and table.

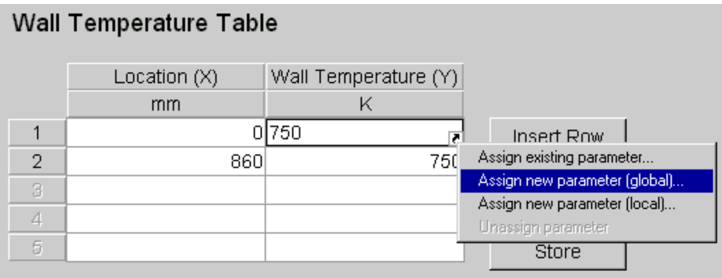


Figure 1-21: Assigning Parameters for Table Values

Wall_T_Up_Catalyst	Exhaust wall temperature at upstream end of pipe downstream of catalyst.	Pipe 32 (Pipe / General / Wall Temperature - Table / Wall temperature at first location in table, 0 mm)
Wall_T_Down_Catalyst	Exhaust wall temperature at downstream end of pipe downstream of catalyst.	Pipe 32 (Pipe / General / Wall Temperature - Table / Wall temperature at last location in table, 860 mm)

Wall_T_Up_Exhaust	Exhaust wall temperature at upstream end of pipe between two exhaust plenums.	Pipe 33 (Pipe / General / Wall Temperature - Table / Wall temperature at first location in table, 0 mm)
Wall_T_Down_Exhaust	Exhaust wall temperature at downstream end of pipe between two exhaust plenums.	Pipe 33 (Pipe / General / Wall Temperature - Table / Wall temperature at last location in table, 970 mm)
Wall_T_Up_Tailpipe	Exhaust wall temperature at upstream end of tail pipe.	Pipe 34 (Pipe / General / Wall Temperature - Table / Wall temperature at first location in table, 0 mm)
Wall_T_Down_Tailpipe	Exhaust wall temperature at downstream end of tail pipe.	Pipe 34 (Pipe / General / Wall Temperature - Table / Wall temperature at last location in table, 330 mm)

6. Copy the data from Cylinder 1 to cylinders 2, 3 and 4. Select **Element**|**Copy data** and select cylinder 1 as the 'Source' and cylinders 2, 3 and 4 as the 'Targets', then select **Apply**.
7. Select **Model**|**Parameters** and the defined parameters for `ottocalc.bwf` are displayed.
8. An additional parameter is required that is not associated with a particular input field to help define the exhaust wall temperatures as a function of the engine speed. To define this parameter select **New Parameter** in the Model Parameters window. In the **Parameter** column for the new item, change the default name `Parameter_1` to `Exhaust_Wall_T_Factor`. Define an initial value of `0.01` in the **Value** column (this will be varied as a function of the case in the case explorer). Define the units of this parameter as `Ratio[-]` in the **Units** column.
9. The following parameters then need to be redefined with a formula that uses the new parameter defined in the previous step.

Parameter	Formula
Exhaust_Wall_1_T	=660+Exhaust_Wall_T_Factor*(980-660)
Exhaust_Wall_2_T	=640+Exhaust_Wall_T_Factor *(950-640)
Exhaust_Wall_3_T	=630+Exhaust_Wall_T_Factor *(930-630)
Wall_T_Up_Catalyst	=550+Exhaust_Wall_T_Factor *(1024-550)
Wall_T_Down_Catalyst	=550+Exhaust_Wall_T_Factor *(965-550)
Wall_T_Up_Exhaust	=510+Exhaust_Wall_T_Factor *(885-510)
Wall_T_Down_Exhaust	=510+Exhaust_Wall_T_Factor *(850-510)
Wall_T_Up_Tailpipe	=480+Exhaust_Wall_T_Factor *(768-480)
Wall_T_Down_Tailpipe	=480+Exhaust_Wall_T_Factor *(737-480)

10. The Model Parameters should then be as shown in the following figure.

Parameter	Type	Value	Unit
AF_Ratio	global	13.34	[-] (Ratio)
Cylinder_Head_T	global	530	K (Temperature)
Duration_of_Combusti	global	47	deg (Angle)
Engine_Speed	global	5000	rpm (Angular Velocity)
Exahust_Wall_2_T	global	=640+Exhaust_Wall_T	K (Temperature)
Exhaust_Port_Wall_T	global	540	K (Temperature)
Exhaust_Wall_1_T	global	=660+Exhaust_Wall_T	K (Temperature)
Exhaust_Wall_3_T	global	=630+Exhaust_Wall_T	K (Temperature)
Exhaust_Wall_T_Factc	global	0.01	[-] (Ratio)
Liner_BDC_T	global	425	K (Temperature)
Liner_TDC_T	global	435	K (Temperature)
Piston_T	global	500	K (Temperature)
Start_of_Combustion	global	-5	deg (Angle)
Vibe_Shape	global	1.6	[-] (Ratio)
Wall_T_Down_Catalys	global	=550+Exhaust_Wall_T	K (Temperature)
Wall_T_Down_Exhaus	global	=510+Exhaust_Wall_T	K (Temperature)
Wall_T_Down_Tailpipe	global	=480+Exhaust_Wall_T	K (Temperature)
Wall_T_Up_Catalyst	global	=550+Exhaust_Wall_T	K (Temperature)
Wall_T_Up_Exhaust	global	=510+E =550+Exhaust_Wall_T_Factor*(1024-550)	
Wall_T_Up_Tailpipe	global	=480+Exhaust_Wall_T	K (Temperature)

Figure 1-22: Model Parameters Window

11. Select **Model**|**Case Explorer** and **Case 1** is shown as the active case as it has a red circle.

12. Select  or **Group**|**Edit** to show the available parameters in the model.

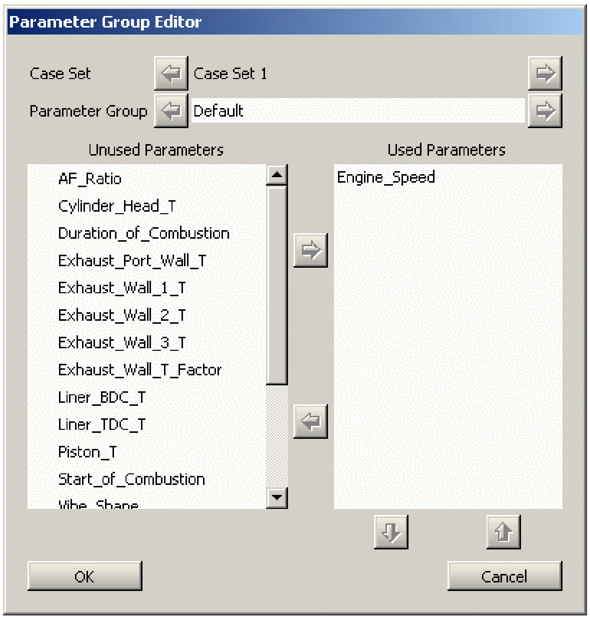


Figure 1-23: Parameter Group Editor Window

13. Select required parameters from the Unused Parameter list and click to display them in the Case Explorer window. In this example **Engine_Speed** is selected first to define it as the main parameter and it is displayed in the first column after **Case Set** in the Case Explorer window. This is relevant for analyzing the results as it will be the default value used for the x-axis for series result plots. Select OK.

14. Reopen the Case Explorer window. Then add some (but not all) of the other parameters to the case explorer as shown in the following figure. This is because the wall temperatures are controlled by the variation of the parameter `Exhaust_Wall_T_factor` which is part of the function that controls the wall temperature value.

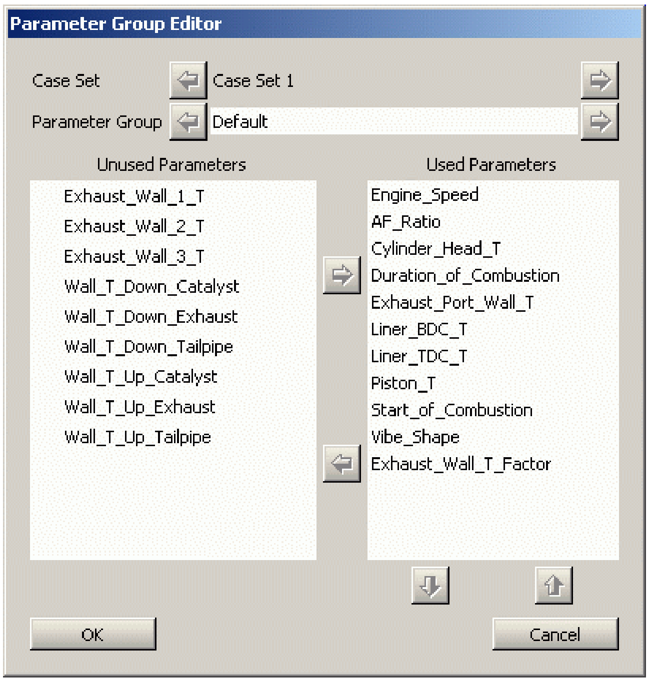


Figure 1-24: Additional Parameters

15. After completing Case 1, select or **Insert | Case** to add another case. Repeat this until there are 17 cases.

16. Enter the values for all cases as shown in the following figures.

17. Double click on a case with the left mouse button to make it active, then select **Model | Parameters** to display the assigned parameters.

Case Set 1	Engine_Speed	AF_Ratio	Cylinder_Head_T	Duration_of_Combustion	Exhaust_Port_Wall_T
	rpm	[-]	K	deg	K
Case 1	7000	12.1	624	52	600
Case 2	6500	12.37	612	51	586
Case 3	6000	12.24	600	50	572
Case 4	5500	12.11	588	49	558
Case 5	5200	12.185	582	48.5	550.5
Case 6	5000	12.26	576	48	543
Case 7	4800	12.42	570	47.5	536
Case 8	4500	12.58	564	47	529
Case 9	4000	13.08	552	46	515
Case 10	3500	13.02	540	45	500
Case 11	3200	12.97	534	44.5	493.5
Case 12	3000	12.92	528	44	487.5
Case 13	2800	13.03	522	43.5	480
Case 14	2500	13.14	516	43	473
Case 15	2000	12.8	504	42	458
Case 16	1500	12.56	492	41	444
Case 17	1000	11.64	480	40	430

Figure 1-25: Case Explorer - Values for Parameters (1)

Liner_BDC_T	Liner_TDC_T	Piston_T	Start_of_Combustion	Vibe_Shape	Exhaust_Wall_T_Factor
K	K	K	deg	[-]	[-]
455	462	570	-6	1.17	1.0
450	456	560	-5	1.2	0.985
440	450	550	-7	1.26	0.974
433	444	540	-8	1.38	0.964
429	441	535	-7.5	1.44	0.9535
425	438	530	-7.5	1.5	0.943
421	435	525	-6	1.53	0.927
417	432	520	-4.5	1.56	0.911
410	426	510	-5	1.62	0.872
400	420	500	-7	1.68	0.762
395	417	495	-7	1.755	0.7035
390	414	490	-6	1.83	0.645
388	411	485	-6	1.955	0.5915
385	408	480	-5	2.08	0.538
375	402	470	-5	2.28	0.304
368	396	460	-3.5	2.38	0.12
360	390	450	5	2.4	0.01

Figure 1-26: Case Explorer - Values for Parameters (2)

1.7.2.2. Run Simulation

1. After saving the model, select **Simulation | Run** to open the following window.

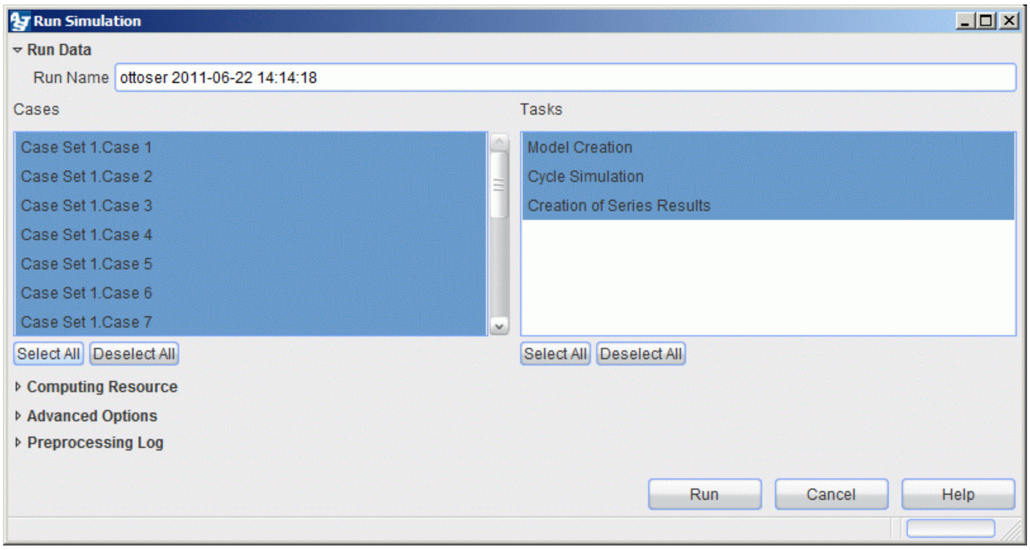


Figure 1-27: Run Simulation Window

2. Select all required cases and tasks. Then select **Run**. The progress of the simulation of each case can be followed in the Simulation Status and Task Information windows**.**

1.7.2.3. Post-processing

After successful completion of the simulation of each case, the series results can be examined in IMPRESS Chart.

1. Select **Simulation│Show Results.**

2. In IMPRESS Chart the series results can be plotted, e.g. volumetric efficiency versus engine speed, as shown in the following figure.

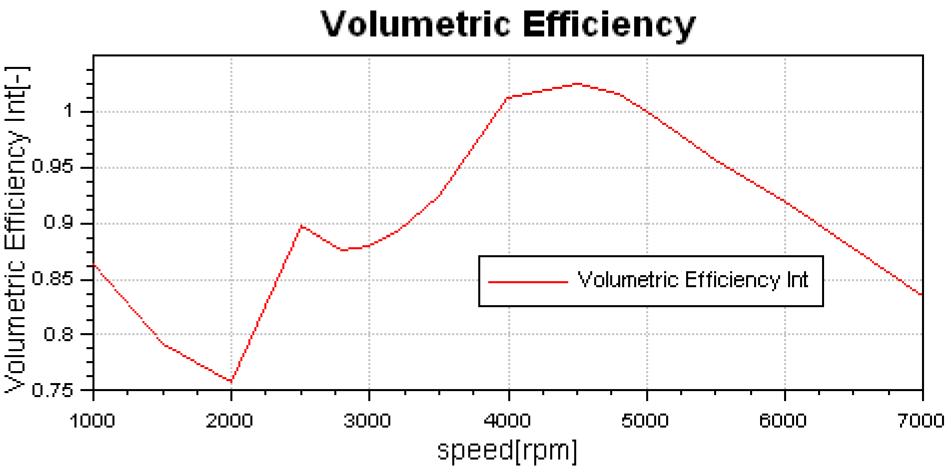


Figure 1-28: Volumetric Efficiency versus Engine Speed