AVL boost

Version 5.1

Examples

January 2008

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

AST Local Support Contact: www.avl.com/ast_support

Revision	**Date**	**Description**	**Document No.**
A	03-May-2002	BOOST v4.0 - Examples	01.0105.0431
B	03-Mar-2003	BOOST v4.0.1 - Examples	01.0105.0436
C	18-Jul-2003	BOOST v4.0.3 - Examples	01.0105.0441
D	23-Jun-2004	BOOST v4.0.4 - Examples	01.0105.0451
E	28-Jan-2005	BOOST v4.0.5 - Examples	01.0105.0462
F	29-Jul-2005	BOOST v4.1 - Examples	01.0105.0467
G	13-Oct-2006	BOOST v5.0 - Examples	01.0105.0500
H	31-Jan-2008	BOOST v5.1 - Examples	01.0105.0510

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.

Table of Contents

List of Figures

1. INTRODUCTION

This manual includes descriptions of BOOST examples. The examples demonstrate some of the main features of the code. The different chapters and sections have also been laid out in a similar manner to the examples directory structure included with a BOOST installation.

1.1. Scope

This manual describes examples of using BOOST to create an engine model. It does not attempt to discuss all the concepts of gas dynamics required to obtain successful solutions. It is the user's responsibility to determine if he/she has sufficient knowledge and understanding of fluid dynamics to apply this software appropriately.

1.2. User Qualifications

This document is a basic qualification for using BOOST and users are recommended to continue with basic and advanced training courses.

1.3. Symbols

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

⚠ **Caution:** Cautions describe conditions, practices or procedures which could result in damage 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.
`monospace`	To indicate a command, a program or a file name, messages, input / output on a screen, file contents or object names.
MENUOPT	A **MENUOPT** font is used for the names of menu options, submenus and screen buttons.

1.4. Configurations

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

1.5. Documentation

BOOST documentation is available in PDF format and consists of the following:

Release Notes
Users Guide
Theory
Primer
Examples
Aftertreatment
Aftertreatment Primer
Linear Acoustics
1D-3D Coupling
Interfaces
Validation
GUI Users Guide
Installation Guide (Windows & UNIX)
Licensing Users Guide
Design Explorer
Python Scripting
Optimization of Multi-body System using AVL Workspace & iSIGHT™
Thermal Network Generator (TNG) User's Guide
Thermal Network Generator (TNG) Primer

2. 4 CYLINDER ENGINE MODEL AT 5000 RPM

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 user is recommended to refer to the **BOOST** Users Guide for more detailed information.

This example is also available using the **General Species Transport** option as:

- `ottocalc_species.bwf`: The setup is identical to the `ottocalc.bwf` except that the general species transport option is used. . See chapters 2.2 and 2.3.1 for details.
- `ottocalc_species_2zone.bwf`: Here the `ottocalc.bwf` is converted to a two-zone model taking into account detailed pollutant formation chemistry in the burned zone. See chapters 2.2 and 2.3.1 for details.

2.1. 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:

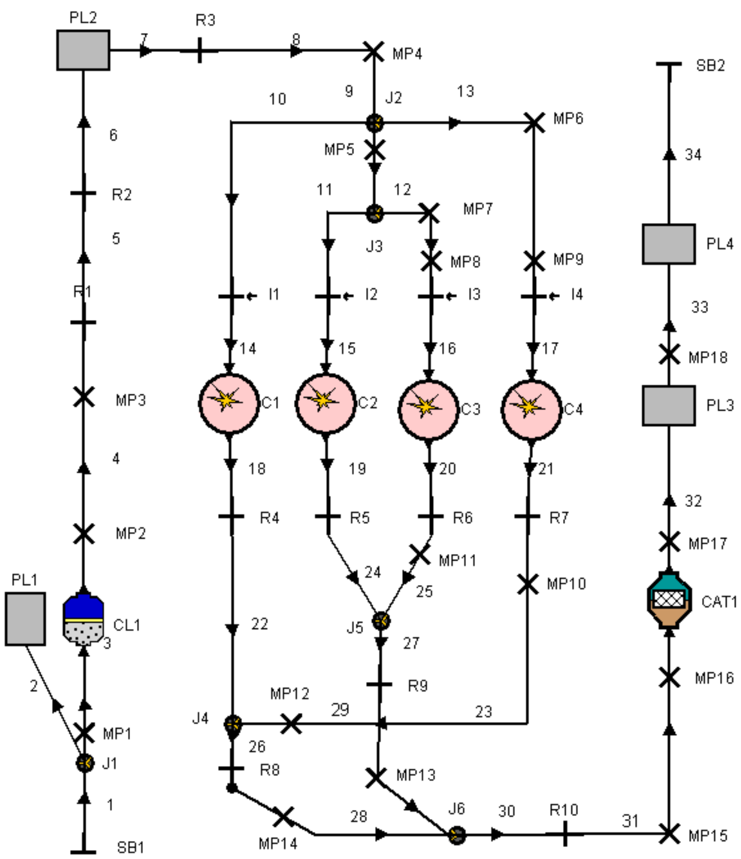


Figure 2—1: Model Schematic for Four Cylinder Engine Model

The model consists of the following elements:

- 4 Cylinders C
- 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.

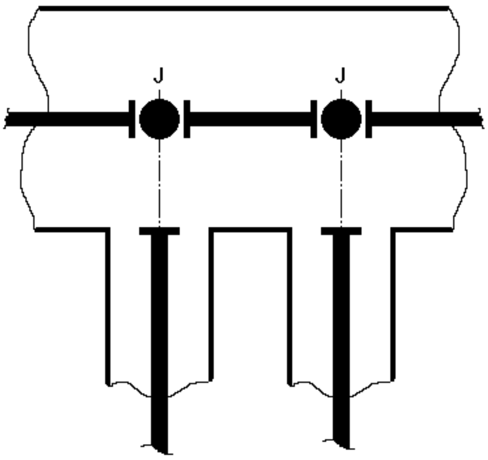


Figure 2—2: Modeling of an Intake Receiver with Pipes and Junctions

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

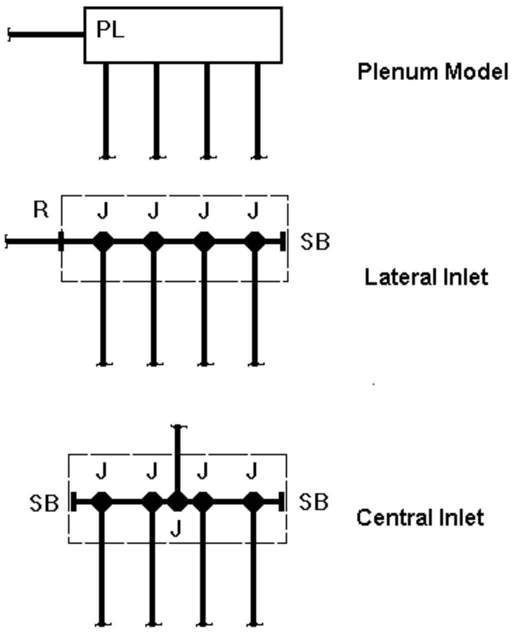


Figure 2—3: Intake Receiver Models

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 2—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.

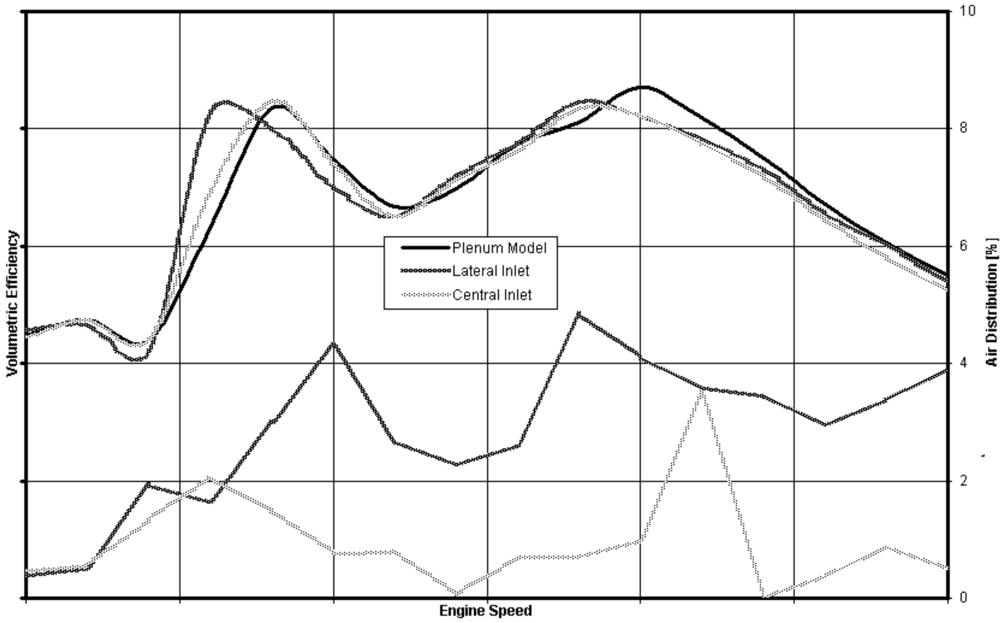


Figure 2—4: Influence of Intake Receiver Modeling on Volumetric Efficiency and Air Distribution

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.

2.2. 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** and select **Cycle Simulation** for **Simulation Tasks**.

1. **GENERAL CONTROL**

Click on the **General Control** sub-group folder in the tree if the window is not displayed.

Species Transport:	Classic
Engine Speed:	5000 rpm
Calculation Mode:	Single
Identical Cylinders	Activate
Mixture Preparation:	External
Fuel:	(Classic Species Transport)
Type:	Gasoline
Lower Heating Value:	43500 kJ/kg
Stoichiometric A/F Ratio:	14.5
Reference Conditions:	
Pressure:	1 bar
Temperature:	24.85 degC

Changes for **General Species Transport:** `ottocalc_species.bwf:`

Species Transport:	General (disables the Fuel Type, Lower Heating Value and Stoichiometric A/F Ratio input)

Click on the **General Species Setup** sub-group folder in the tree if the window is not displayed.

Species:	GASOLINE, O2, N2, CO2, H2O, CO, H2, O, NO
Key, Chemistry:	empty
Fuel:	GASOLINE
User Database:	disabled

Changes for **General Species Transport:** `ottocalc_species_2zone.bwf:`

Species Transport:	General (disables the Fuel Type, Lower Heating Value and Stoichiometric A/F Ratio input)

Click on the **General Species Setup** sub-group folder in the tree if the window is not displayed.

Species:	GASOLINE, O2, N2, CO2, H2O, CO, H2, O, OH
	H, HO2, H2O2, N, NO, N2O
Key, Chemistry:	brnd, avlchembrnd.inp
Fuel:	GASOLINE
User Database:	disabled

Engine Speed

The engine speed is the revolution speed of the crankshaft. For steady state simulations, it is kept constant. For transient simulations it is the starting value and is kept constant for the first three cycles to dampen excessive gas dynamics due to the initialization. Afterwards, the instantaneous engine speed is calculated from solving the moment of momentum equation applied to the crankshaft at each time step.

Reference Conditions

The reference conditions (pressure and temperature) are required in order to calculate specific engine performance data such as delivery ratio, volumetric efficiency etc. related to ambient conditions. It is the user's responsibility to ensure that these conditions match the conditions at the system boundary from which the engine aspirates its air. Otherwise, the results might be misleading.

2. TIME STEP CONTROL

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

Cycle:	4-Stroke
Maximum Calculation Period:	
Engine Cycles:	50
Pipes:	
Average Cell Size:	30 mm
Traces Saving Interval:	5 deg
Restart:	
Restart Data Saving Interval:	720 deg
Convergence Control:	Activate

Maximum Calculation Period

The maximum calculation period sets the crank angle interval after which the simulation stops and the results will be written to the `.bst` file. For steady state simulations, it has to be sufficiently long in order to achieve stable calculation results. It is recommended to use a multiple of the cycle duration. The required calculation period until stable conditions are achieved depends upon the engine configuration.

With an increasing number of cylinders, the calculation period may become shorter. 4-stroke engines need shorter calculation periods than 2-stroke engines. For turbocharged (TC) engines, especially if the boost pressure is calculated from the turbine size, significantly longer calculation periods than for naturally aspirated (NA) engines are required.

For an initial estimate, the following data may be used:

- Single cylinder NA 4-stroke engine: 7200 degrees CRA
- Multi cylinder NA 4-stroke engine: 4320 degrees CRA
- Multi cylinder TC 4-stroke engine: 14400 degrees CRA
- Single cylinder 2-stroke engine: 7200 degrees CRA
- Multi cylinder 2-stroke engine: 4320 degrees CRA

Since this engine is a multi-cylinder NA 4 stroke engine, 14400 degrees CRA should be more than sufficient to achieve steady state conditions.

Pipes

The user can either specify the calculation in time step in degree crankangles or a target cell size in mm. From the stability criterion for the pipe flow and from the input time step or target cell size, **BOOST** will calculate the required cell size or the required time step respectively.

The time step for the calculation determines the accuracy (especially the frequency resolution) of the calculation result. However, the number of cells in the pipe system increases dramatically with decreasing time step, which increases the required CPU time. In order to avoid unnecessarily large output files, a separate time step for the saving of the results (time step for traces and animation output) has to be specified.

Restart Data Saving Interval

A restart data saving interval may be specified in order to save restart data at regular crank angle intervals. No restart data will be written to the hard disk if the restart data saving interval is 0 degrees CRA.

Restart

The restart capability of **BOOST** allows an already completed calculation to be continued.

If **Restart** is activated, the new calculation starts with initial conditions taken from a restart file. For a single calculation, the maximum calculation period is the sum of the calculation period of the initial calculation and that of the restart calculation.

If **Restart** is deactivated, the new calculation starts with the initial values specified in the data set.

There are two restart files with the extension .rs0 and .rs1. The first restart file is written to file name .rs0, the second to file name .rs1. The third restart file is written again to file name .rs0. Thus, only the last and second to last restart file exist.

In case of a restart, the program checks first which of the two restart files is the younger one and takes the conditions stored on this file for the initialization. If just one restart file exists, this one will be used.

If neither filename.rs0 nor filename.rs1 exist, the program run will be interrupted with an error message.

Time Reset

To avoid long transient output, activate **Time Reset.** This results in, in the event of a restart, that only the transient results from the restart on will be written to the .bst-file. The transient results from the calculation from which the restart file was obtained will be lost. If **Time Reset** is deactivated, the complete history will again be stored on the .bst-file and can be analyzed using the transient analysis feature of the **BOOST** post-processor.

3. FIRING ORDER

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

Firing Interval (deg)
0
540
180
360

4. 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

5. **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

6. **ENGINE FRICTION**

Click on the **Engine Friction** sub-group folder and select **Table** and enter `1` for **Friction Multiplier**.

Click on the **Engine Friction[1]: friction_list** sub-group folder and enter the following data:

Load: 10 bar

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

Select **Apply** and the sub-group icon turns green to confirm that valid data has been specified.

2.3. 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**.

2.3.1. 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

BOOST offers three scavenging models: Perfect Mixing, Perfect Displacement, and User-defined scavenging model. Perfect Mixing is used, in which the gas flowing into the cylinder is mixed immediately with the cylinder contents. The gas leaving the cylinder has the same composition as the mixture in the cylinder. The perfect mixing model is the standard scavenging model for the simulation of 4-stroke engines.

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.

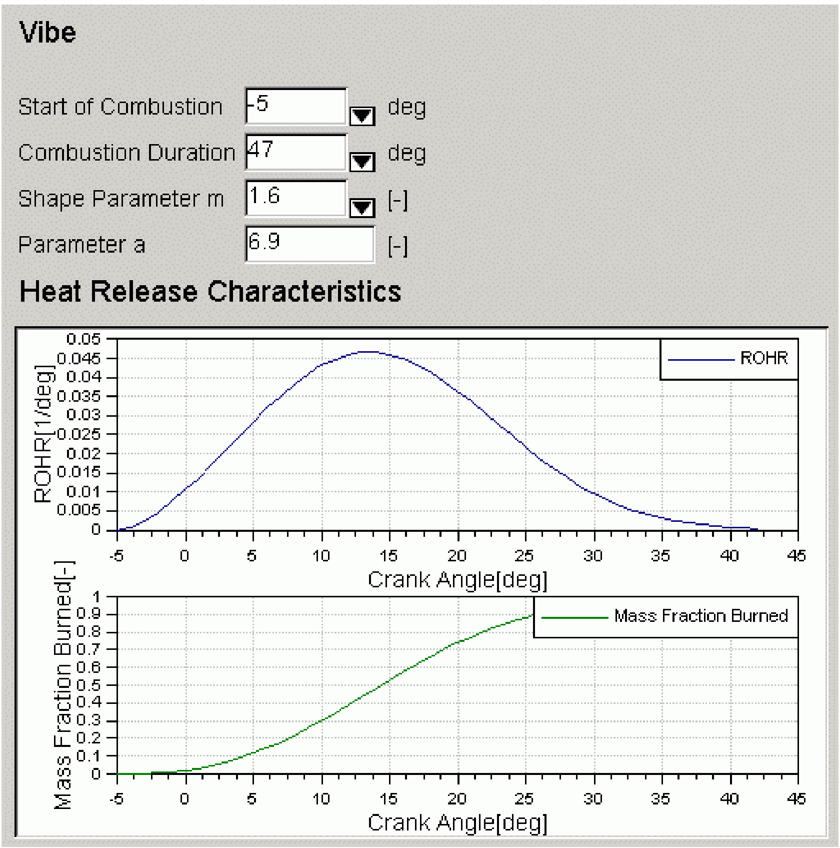


Figure 2—5: Vibe Window

Additional settings for **General Species Transport:** `ottocalc_species.bwf`**:**

Single Zone Chemistry:	disabled
Gas Exchange Phase Chemistry:	disabled
Solver absolute Tolerance:	disabled
Solver relative Tolerance:	disabled

Additional settings for **General Species Transport:** `ottocalc_species_2zone.bwf`**:**

Single Zone Chemistry:	enabled; select *brnd*
Gas Exchange Phase Chemistry:	disabled
Two Zone Unburned Chemistry:	disabled
Two Zone Burned Chemistry:	enabled; select *brnd*
Solver absolute Tolerance:	disabled
Solver relative Tolerance:	disabled

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 mm²**	**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.

Click on the **VPS [1]: Pipe 14 Intake** sub-group folder and then click on **Valve Controlled** to access the following input fields:

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

Click on **Lift Curve** to open the following window and enter the relevant data:

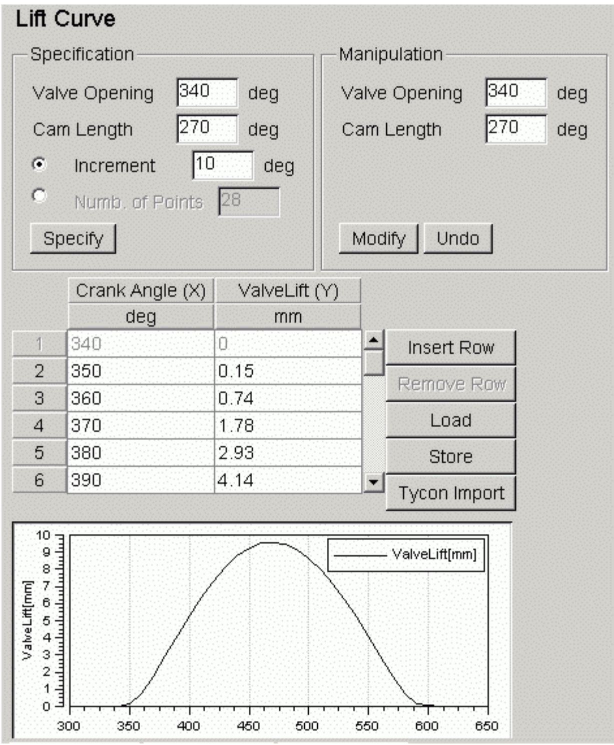


Figure 2—6: Lift Curve Window

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

Refer to the Table on page 2-14 for **Crank Angle** and **Valve Lift** input data for the Intake Pipe 14 Lift Curve.

Click on **Flow Coefficient** to open the following window and enter the relevant data:

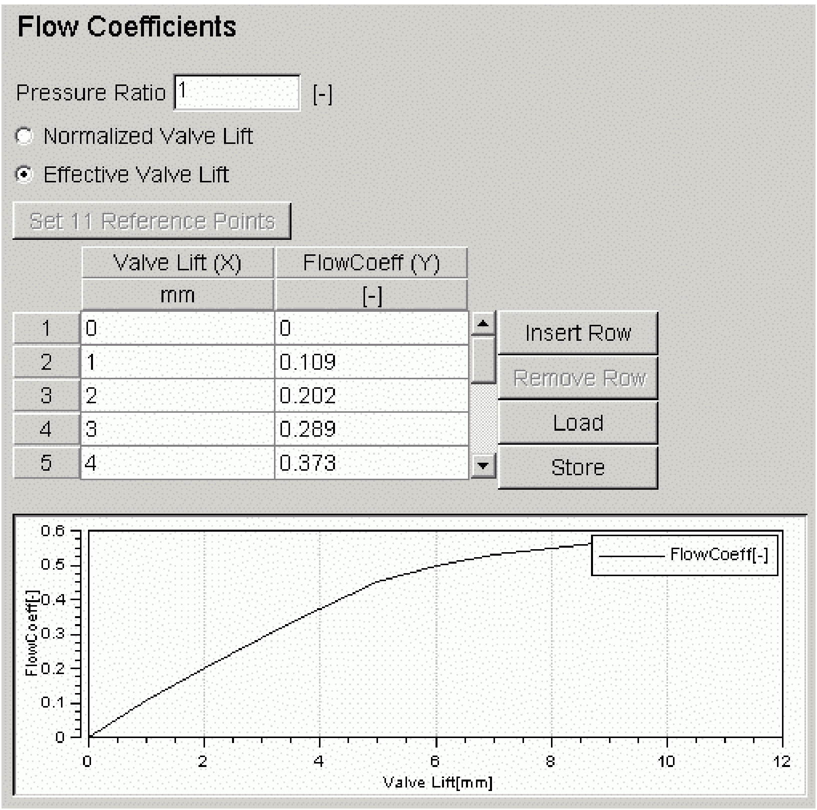


Figure 2—7: Flow Coefficient Window

Pressure Ratio	1
Effective Valve Lift	Selected

Refer to the following Table for **Valve Lift** and **Flow Coefficient** input data for the Intake Pipe 14 Flow Coefficient.

Click on the **VPS [2]: Pipe 18 Exhaust** sub-group folder and then click on **Valve Controlled** to access the following input fields:

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

Click on **Lift Curve** and enter the relevant data:

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

Refer to the following Table for **Crank Angle** and **Valve Lift** input data for the Exhaust Pipe 18 Lift Curve.

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

Pressure Ratio	1
Effective Valve Lift	Selected

Refer to the following Table for **Valve Lift** and **Flow Coefficient** input data for Exhaust Pipe 18 Flow Coefficient.

Intake Pipe 14				**Exhaust Pipe 18**			
Lift Curve		**Flow Coefficient**		**Lift Curve**		**Flow Coefficient**	
Crank Angle (X) deg	Valve Lift (Y) mm	Valve Lift (X) mm	Flow Coeff (Y)	Crank Angle (X) deg	Valve Lift (Y) mm	Valve Lift (X) mm	Flow Coeff (Y)
340	0	0	0	130	0	0	0
350	0.15	1	0.109	140	0.2	1	0.149
360	0.74	2	0.202	150	0.91	2	0.29
370	1.78	3	0.289	160	1.98	3	0.475
380	2.93	4	0.373	170	3.21	4	0.615
390	4.14	5	0.453	180	4.39	5	0.664
400	5.28	6	0.498	190	5.53	6	0.684
410	6.37	7	0.529	200	6.54	7	0.693
420	7.31	8	0.551	210	7.44	8	0.697
430	8.12	9	0.567	220	8.17	9	0.7
440	8.77	10	0.579	230	8.71	10	0.702
450	9.25	11	0.584	240	9.07	11	0.703
460	9.52			250	9.32		
470	9.58			260	9.32		
480	9.44			270	9.01		
490	9.13			280	8.58		
500	8.63			290	7.94		
510	7.96			300	7.06		
520	7.12			310	6.05		
530	6.2			320	5.04		
540	5.25			330	3.92		
550	4.12			340	2.78		
560	2.94			350	1.42		
570	1.76			360	0.72		
580	0.75			370	0.18		
590	0.17			380	0.06		
600	0.05			390	0		
610	0						

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

- A pipe is connected to each valve (Figure 2—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 2—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.

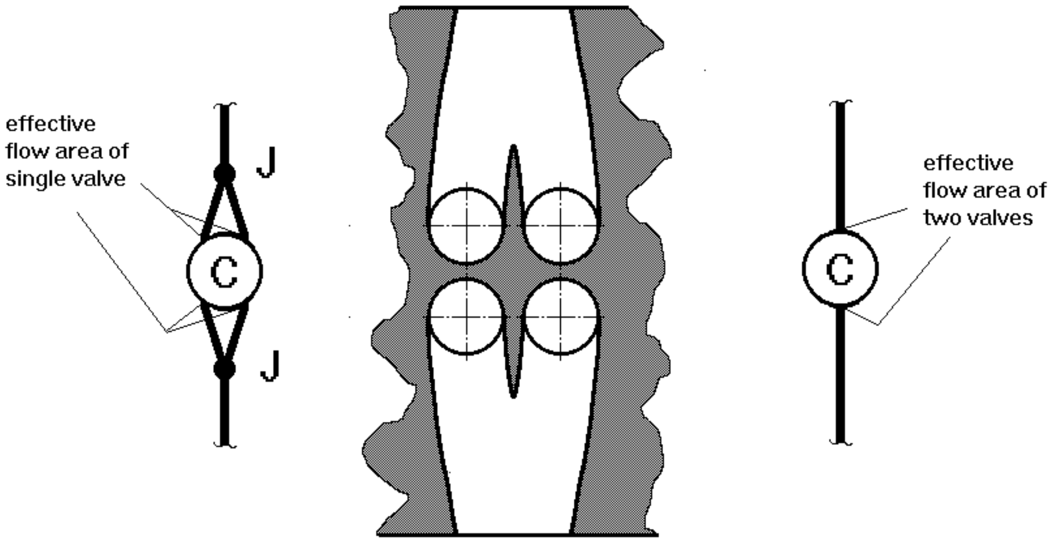


Figure 2—8: Modeling multi-valve engines

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:

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

The flow coefficients themselves are defined in the box appearing after pressing Flow Coefficients. Note that the as Flow Coefficients are specified just for one pressure ratio, they will be used irrespective of the actual pressure ratio across the valves.

2.3.2. 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.

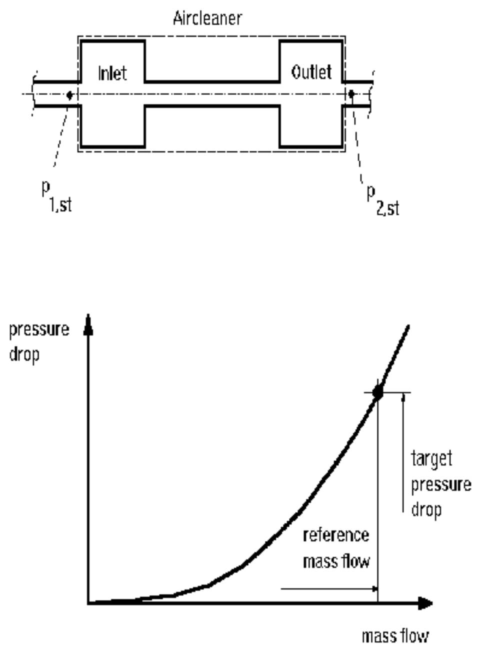


Figure 2—9: Steady State Air Cleaner Performance

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

2.3.3. 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)	0 1/m

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

2.3.4. 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.

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 choose "Continuous" as injection method.

2. **MASS FLOW**

Click on the **Mass Flow** sub-group folder, choose "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. **SPECIES OPTIONS**

Click on the **Species Options** sub-group folder, choose "Fuel" and deactivate "Consider Heat of Evaporation"

4. **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

2.3.5. 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 2.2. – 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

2.3.6. 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.

2.3.7. 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.

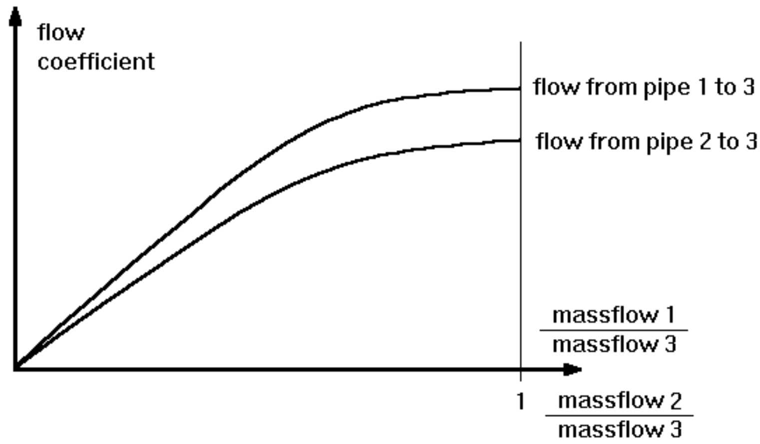


Figure 2—10: Flow Coefficients of a Junction

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 were 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:

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

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

2.3.8. 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

2.3.9. 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.

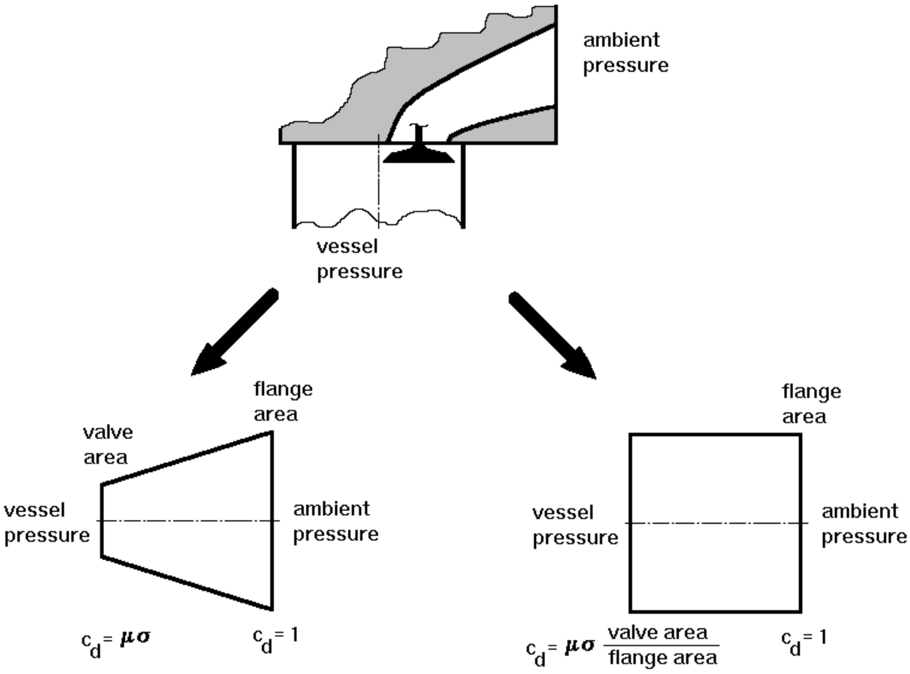


Figure 2—11: Flow Coefficient Measurement

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 2—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.

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

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	0 110	55 44				
Pipe 3	0 220	44 80				
Pipe 4	0 110 220	70 60 60				
Pipe 8	0 52.5 105	75 75 65				
Pipe 9	0 80	65 55				
Pipes 10 - 13	0 70 115 170 225 265 320	45 42.7 41.3 36.8 33.5 33.4 33.4	0 105 210 315	0 120 60 10000		
Pipe 22					0 102.5 213.75 305	0.04 0.04 0.023 0.023
Pipe 23					0 102.5 213.75 285	0.04 0.04 0.023 0.023
Pipe 24					0 110 205 300	0.04 0.04 0.023 0.023
Pipe 25					0 115 182.5 270	0.04 0.04 0.023 0.023
Pipe 31	0 220 400 570 970	46 46 45 46 46	0 220 570 970	0 10000 100 10000	0 220 570 970	0.019 0.06 0.06 0.019

2.3.10. 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

2.3.11. 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.

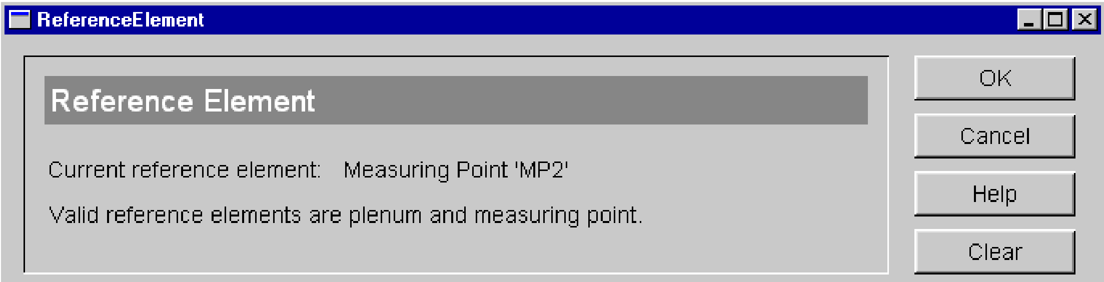


Figure 2—12: Reference Point for Volumetric Efficiency

Note: Save the model before starting calculation.

2.4. 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. Select **View Logfile** to view more detailed information on the simulation run produced by the simulation kernel.

Once the job is complete select **Close** to exit.

2.5. 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.

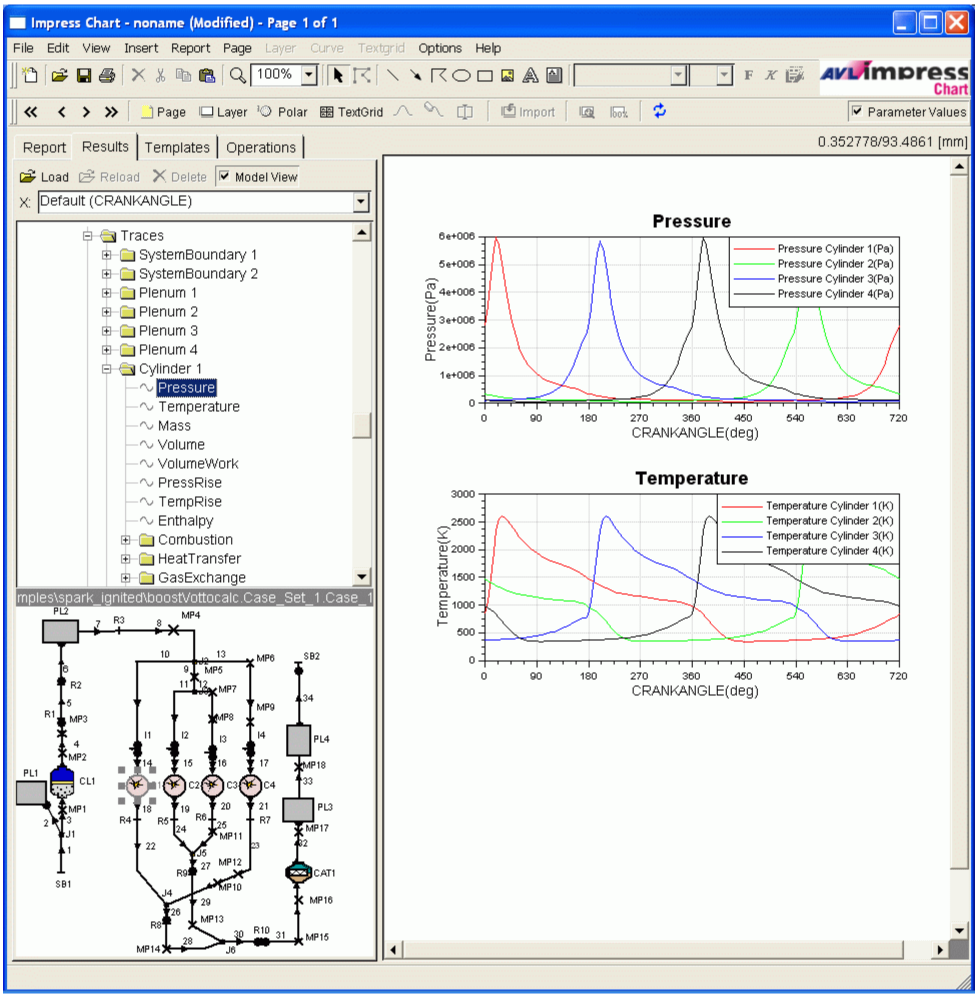


Figure 2—13: Results Window

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.

3. CASE SERIES CALCULATION

Using the `ottocalc.bwf` single case model created in Chapter 2, 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.

The user is recommended to refer to Chapter 3.5 of the **BOOST** Users Guide for more detailed information on the case series calculation.

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

3.1. Assign New Parameters

Firstly the parameters must be set for Case 1.

1. Select **Simulation**|**Control** to open the **General Controls** 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.

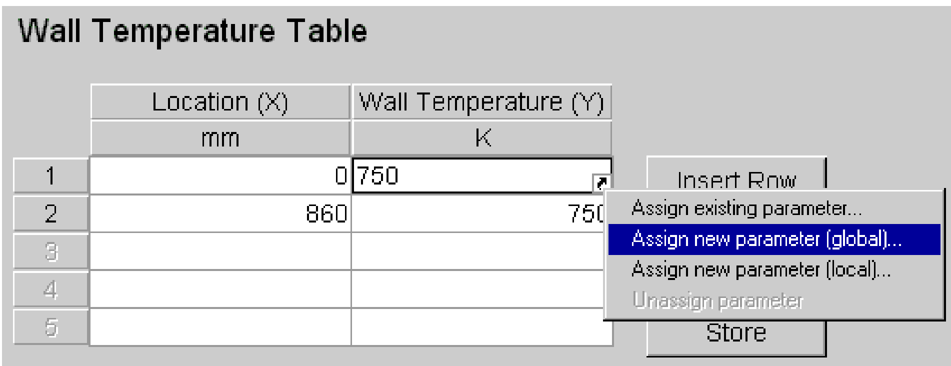


Figure 3—1: Assigning Parameters for Table Values

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_Fact	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 3—2: 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.

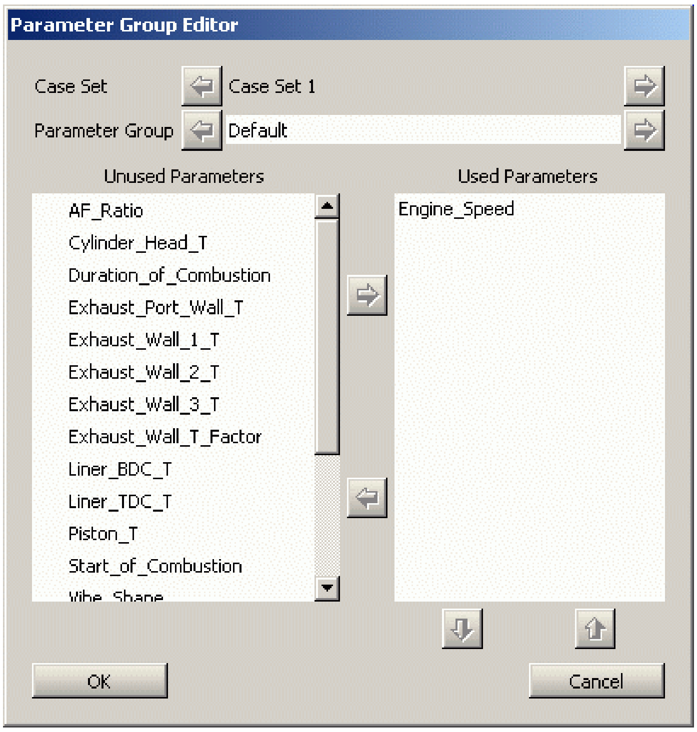


Figure 3—3: Parameter Group Editor Window

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.

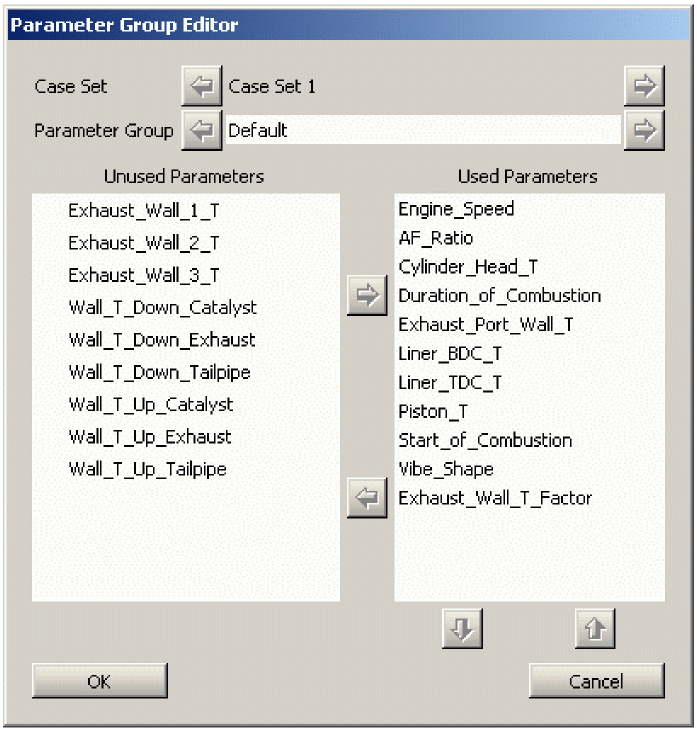


Figure 3—4: Additional Parameters

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

Figure 3—5: Case Explorer - Values for Parameters (1)

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

Figure 3—6: Case Explorer - Values for Parameters (2)

☞ **Note:** Save the model before starting calculation.

3.2. Run Simulation

Select **Run**|**Case Set** to open the following window.

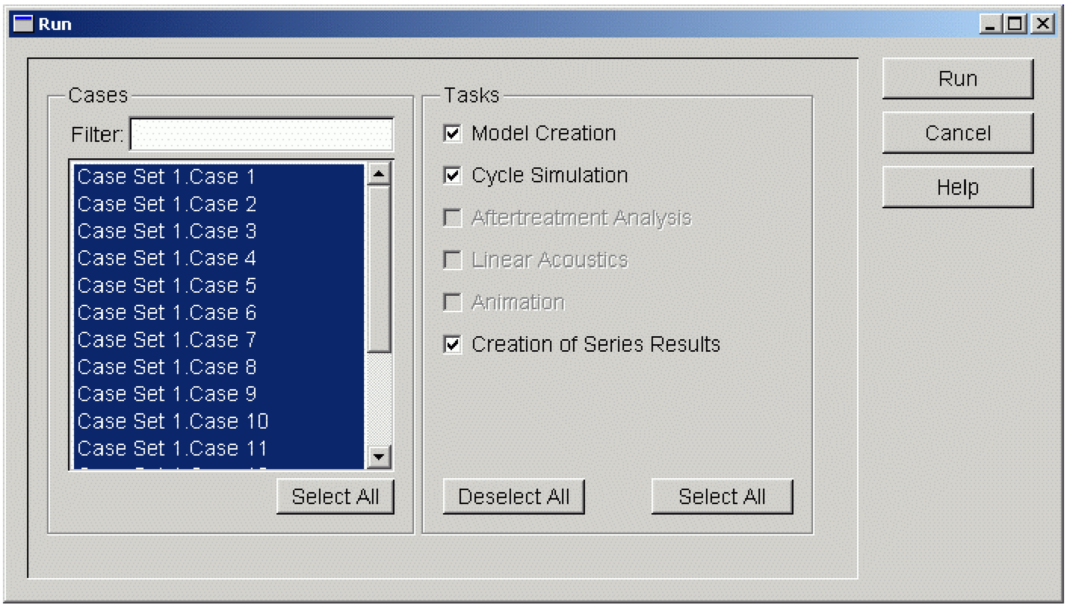


Figure 3—7: Run Simulation Window

1. Select **Model Creation, Cycle Simulation** and **Creation of Series Results**.
2. Select **Run**.

The progress of the simulation of each case can be followed from **Model|Case Explorer.**

3.3. Post-processing

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

1. Select **Simulation|Create Series Results.**
2. Select **Simulation|Show Results.**
3. In IMPRESS Chart the series results can be plotted, e.g. volumetric efficiency versus engine speed, as shown in the following figure.

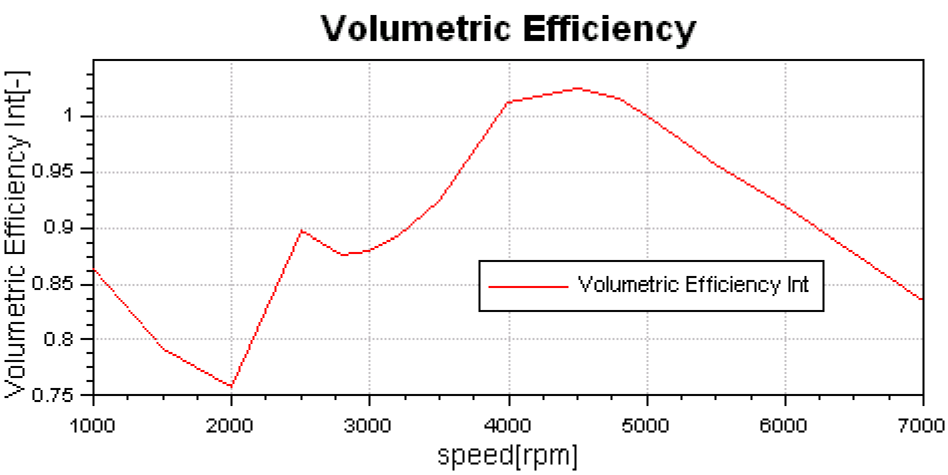


Figure 3—8: Volumetric Efficiency versus Engine Speed

4. DIESEL ENGINE WITH TURBOCHARGER

This chapter describes how to create and run the model of a 6 cylinder diesel engine with turbocharger running at 2500 rpm WOT. The `tcicalc.bwf` file is used for this example and is located in the Compression_ignited folder.

The user is recommended to refer to the **BOOST** Users Guide for more detailed information.

This example is also available using the **General Species Transport** option as:

- `tcicalc_species.bwf`: The setup is identical to the `tcicalc.bwf` except that the general species transport option is used. See chapters 4.2 and 4.3.1 for details.

4.1. 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:

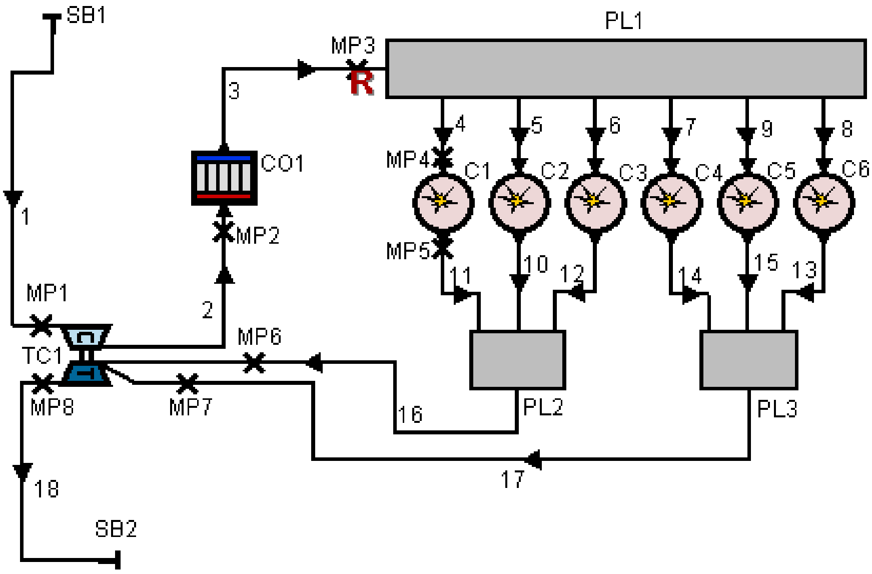


Figure 4—1: Diesel Engine with Turbocharger Model

This model consists of the following:

• 6 Cylinders	C
• 1 Turbocharger	TC
• 1 Air Cooler	CO
• 3 Plenums	PL
• 2 System Boundaries	SB
• 8 Measuring Points	MP
• 18 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 Connection to insert a pipe and attach it to the required elements by clicking on the activated circles (triangles for the cylinder).

4.2. 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** and select **Cycle Simulation** for **Simulation Tasks**.

1. **GENERAL CONTROL**

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

Engine Speed:	2500 rpm
Calculation Mode:	Single
Identical Cylinders	Activate
Mixture Preparation:	Internal
Fuel:	
Type:	Diesel
Lower Heating Value:	42800 kJ/kg
Stoichiometric A/F Ratio:	14.7
Reference Conditions:	
Pressure:	1 bar
Temperature:	24.85 degC

2. **TIME STEP CONTROL**

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

Cycle:	4-Stroke
Maximum Calculation Period:	
Degree Crankangle:	10800 deg
Pipes:	
Calculation Step Size:	1 deg
Traces Saving Interval:	3 deg
Restart:	
Restart Data Saving Interval:	720 deg

3. FIRING ORDER

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

Firing Interval (deg)

0

480

240

600

360

120

4. INITIALIZATION

Click on the **Initialization** sub-group folder and select **A/F-Ratio** for **Ratio**. Select **Add Set** and enter the following data in the input fields.

	Pressure (bar)	Gas Temp (degC)	Fuel Vapor	Combustion Products	Ratio Value
Set 1	2.6	66.85	0	0	14.7
Set 2	2.4	596.85	0	0.5413793	14.7

5. ENGINE FRICTION

Click on the **Engine Friction** sub-group folder, select **Table** and enter `1` for **Friction Multiplier**.

Click on the **Engine Friction[1]: friction_list** sub-group folder and enter the following data:

Load: 15 bar

Engine Speed (X) rpm	FMEP (Y) bar
800	0.7
2700	2.5

Select **Apply** and the sub-group icon turns green to confirm that valid data has been specified.

4.3. 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.

4.3.1. Cylinder

The data for the cylinder is listed in the following table. 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:	100 mm
Stroke:	130 mm
Compression Ratio:	18
Con-rod Length:	220 mm
Piston Pin Offset:	0 mm
Effective Blow by Gap:	0 m
Mean Crankcase Press:	1 bar
Scavenge Model:	Perfect Mixing

2. **INITIALIZATION**

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

Initial Conditions at EO (Exhaust Valve Opening)

Pressure:	4.5 bar
Temperature:	726.85 degC

Initial Gas Composition

Ratio Type:	A/F Ratio
Ratio Value:	14.7
Fuel Vapour:	0
Combustion Products:	0.628

3. **COMBUSTION**

Click on the **Combustion** sub-group folder. Select **Vibe** from the pull down menu for **Heat Release** and enter `0.0001` kg for **Fuel Mass / Cycle**.

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

Start of Combustion:	714 deg
Combustion Duration:	75 deg
Shaping Parameter m	0.85

Parameter a 6.9

Additional settings for **General Species Transport:** `tcicalc_species.bwf`**:**

Single Zone Chemistry:	disabled
Gas Exchange Phase Chemistry:	disabled
Solver absolute Tolerance:	disabled
Solver relative Tolerance:	disabled

4. HEAT TRANSFER

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

Cylinder:	Woschni 1978	
Ports:	None	
Piston:		
Surface Area:	14000 mm^2	
Wall Temperature:	276.85 degC	
Piston Calibration Factor:	1	
Cylinder Head:		
Surface Area:	11000 mm^2	
Wall Temperature:	246.85 degC	
Head Calibration Factor:	1	
Liner:		
Surface Area:	500 mm^2	(Piston at TDC)
Wall Temperature:	176.85 degC	(Piston at TDC)
Wall Temperature:	86.85 degC	(Piston at BDC)
Liner Calibration Factor:	1	
Combustion System	DI	
Incylinder Swirl Ratio:	1.9	

5. VALVE PORT SPECIFICATIONS

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

Controlled by	
Pipe	**Control**
4	Valve
11	Valve

Click on the **VPS [1]: Pipe 4 Intake** sub-group folder and then click on **Valve Controlled** to access the following input fields:

Inner Valve Seat (=Reference) Diameter	41 mm
Valve Clearance	0.3 mm
Scaling Factor for Eff. Flow Area	1

Click on **Lift Curve** and enter the relevant data:

Specification		Manipulation	
Valve Opening	310 deg	Valve Opening	310 deg
Cam Length	310 deg	Cam Length	310 deg
Increment	10 deg		

Refer to the following Table for **Crank Angle** and **Valve Lift** input data for the Intake Pipe 4 Lift Curve.

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

Pressure Ratio	0.99
Effective Valve Lift	Active

Refer to the following Table for **Valve Lift** and **Flow Coefficient** input data for the Intake Pipe 4 Flow Coefficient.

Click on the **VPS [2]: Pipe 11 Exhaust** sub-group folder and then click on **Valve Controlled** to access the following input fields:

Inner Valve Seat (=Reference) Diameter	39 mm
Valve Clearance	0.4 mm
Scaling Factor for Eff. Flow Area	1

Click on **Lift Curve** and enter the relevant data:

Specification		Manipulation	
Valve Opening	82 deg	Valve Opening	82 deg
Cam Length	340 deg	Cam Length	340 deg
Increment	10 deg		

Refer to the following Table for **Crank Angle** and **Valve Lift** input data for the Exhaust Pipe 11 Lift Curve.

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

Pressure Ratio	0.99
Effective Valve Lift	Selected

Refer to the following Table for **Valve Lift** and **Flow Coefficient** input data for Exhaust Pipe 11 Flow Coefficient.

Intake Pipe 4				Exhaust Pipe 11			
Lift Curve		**Flow Coefficient**		**Lift Curve**		**Flow Coefficient**	
Crank Angle (X) deg	Valve Lift (Y) mm	Valve Lift (X) mm	Flow Coeff (Y)	Crank Angle (X) deg	Valve Lift (Y) mm	Valve Lift (X) mm	Flow Coeff (Y)
310	0	0	0	82	0	0	0
320	0.04	1	0.085	92	0.03	1	0.095
330	0.14	2	0.155	102	0.14	2	0.183
340	0.26	3	0.22	112	0.26	3	0.25
350	0.45	4	0.285	122	0.38	4	0.33
360	1.02	5	0.345	132	0.56	5	0.393
370	2.16	6	0.4	142	1.13	6	0.47
380	3.7	7	0.438	152	2.25	7	0.53
390	5.31	8	0.465	162	3.76	8	0.603
400	6.81	9	0.485	172	5.38	9	0.65
410	8.11	10	0.497	182	6.9	10	0.69
420	9.21	11	0.5	192	8.26	11	0.71
430	10.08	12	0.505	202	9.42	12	0.735
440	10.72			212	10.36		
450	11.13			222	11.09		
460	11.3			232	11.61		
470	11.23			242	11.9		
480	10.91			252	11.96		
490	10.37			262	11.81		
500	9.58			272	11.43		
510	8.58			282	10.83		
520	7.35			292	10.01		
530	5.93			302	8.98		
540	4.35			312	7.74		
550	2.75			322	6.31		
560	1.41			332	4.74		
570	0.61			342	3.13		
580	0.31			352	1.74		
590	0.19			362	0.84		
600	0.07			372	0.46		
610	0.01			382	0.03		
620	0			392	0.21		
				402	0.09		
				412	0.01		
				422	0		

4.3.2. Air Cooler

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

1. **GENERAL**

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

Geometrical Properties

Total Air Cooler Volume:	10 (l)
Inlet Collector Volume:	3 (l)
Outlet Collector Volume:	3 (l)
Length of Cooling Core:	600 mm

Gas Properties

Mass Flow	0.333 kg/s
Target Pressure Drop	5000 Pa
Target Outlet Temperature	340 K
Inlet Air Temperature	415 K
Inlet Pressure	260000 Pa
Coolant Temperature	298 K

2. **FLOW COEFFICIENTS**

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

Pipe 2 Inflow	1	**Pipe 2 Outflow**	1
Pipe 3 Inflow	1	**Pipe 3 Outflow**	1

4.3.3. Turbocharger

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

1. **GENERAL**

Click on the **General** sub-group folder and select the calculation type:

Calculation Type: Simplified Model

2. **SIMPLIFIED MODEL**

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

Calculation Mode:	Turbine Layout Calculation
Equiv. Turbine Coeff.	0.15
Turbo Charger overall Efficiency	0.54
Compressor Efficiency	0.79
Mechanical Efficiency	0.98
Compressor Pressure Ratio	2.6
Inlet Interference Flow coefficient	0.15

4.3.4. System Boundary

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

1. **BOUNDARY CONDITIONS**

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

Select **Local Boundary Conditions**.

	Preference	Pressure (Pa)	Gas Temp (K)	Ratio Type	Ratio Value
SB 1	Set 1	100000	298	A/F Ratio	10000
SB 2	Set 1	100000	780	A/F Ratio	28

2. **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 18 Inflow**	0.75	**Pipe 18 Outflow**	0.75

4.3.5. Plenum

The data for the plenums is listed in the following table. 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**. Then select the relevant set from the **Preferences** pull-down menu.

	Volume (l)	Initialization
Plenum 1	6	Set 1
Plenum 2	0.3	Set 2
Plenum 3	0.3	Set 2

2. **FLOW COEFFICIENTS**

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

Plenum 1	**Pipe 3 Inflow**	0.95	**Pipe 3 Inflow**	0.95
	Pipe 4 Inflow	0.9	**Pipe 4 Inflow**	0.9
	Pipe 5 Inflow	0.9	**Pipe 5 Inflow**	0.9
	Pipe 6 Inflow	0.9	**Pipe 6 Inflow**	0.9
	Pipe 7 Inflow	0.9	**Pipe 7 Inflow**	0.9
	Pipe 8 Inflow	0.9	**Pipe 8 Inflow**	0.9
	Pipe 9 Inflow	0.9	**Pipe 9 Inflow**	0.9
Plenum 2	**Pipe 10 Inflow**	0.9	**Pipe 10 Inflow**	0.9
	Pipe 11 Inflow	0.9	**Pipe 11 Inflow**	0.9
	Pipe 12 Inflow	0.9	**Pipe 12 Inflow**	0.9
	Pipe 16 Inflow	0.95	**Pipe 16 Inflow**	0.95
Plenum 3	**Pipe 13 Inflow**	0.9	**Pipe 13 Inflow**	0.9
	Pipe 14 Inflow	0.9	**Pipe 14 Inflow**	0.9
	Pipe 15 Inflow	0.9	**Pipe 15 Inflow**	0.9
	Pipe 17 Inflow	0.95	**Pipe 17 Inflow**	0.95

4.3.6. Pipes

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.

Global Initialization is used for pipes 3 - 17 and Local Initialization is used for pipes 1, 2 and 18.

1. **GENERAL**

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

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

	Pipe Length (mm)	Diameter (mm)	Friction Coeff.	Heat Transfer Factor	Wall Temp (K)	Initial-ization
Pipe 1	400	85	0.02	1	298	see below
Pipe 2	800	85	0.02	1	350	see below
Pipe 3	800	85	0.02	1	320	Set 1
Pipes 4 - 9	700	TABLE	0.02	1	TABLE	Set 1
Pipe 10	250	38	0.02	1	500	Set 2
Pipe 11	300	38	0.02	1	500	Set 2
Pipe 12	300	38	0.02	1	500	Set 2
Pipe 13	250	38	0.02	1	500	Set 2
Pipe 14	300	38	0.02	1	500	Set 2
Pipe 15	300	38	0.02	1	500	Set 2
Pipe 16	300	40	0.02	1	470	Set 2
Pipe 17	600	40	0.02	1	470	Set 2
Pipe 18	1500	100	0.02	1	450	see below

The diameter and wall temperature data for pipes 4 - 9 is listed in the following table. Click on ▼ and then select the **Table** button which appears on the input field to open the input window. Select **Insert Row** to activate the input fields.

	Diameter Table		Wall Temperature Table	
	Location X (mm)	Diameter Y (mm)	Location X (mm)	Wall Temp Y (K)
Pipes 4 - 9	0	55	0	320
	100	44	580	320
	580	38	700	400
	700	41		

2. **INITIALIZATION**

For the following pipes, select **Local Initialization** and enter the data shown.

	Preference	Pressure (Pa)	Gas Temp (K)	Ratio Type	Ratio Value
Pipe 1	Set 1	100000	298	A/F Ratio	10000
Pipe 2	Set 1	260000	415	A/F Ratio	10000
Pipe 18	Set 1	1050000	780	A/F Ratio	28

4.3.7. Measuring Point

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	400	Extended
Measuring Point 2	800	Extended
Measuring Point 3	800	Extended
Measuring Point 4	700	Extended
Measuring Point 5	0	Extended
Measuring Point 6	300	Extended
Measuring Point 7	600	Extended
Measuring Point 8	0	Extended

4.3.8. Reference Point for Volumetric Efficiency

Select **Simulation**|**Volumetric Efficiency** to open the following window. In this example select **Measuring Point 3** as the reference element.

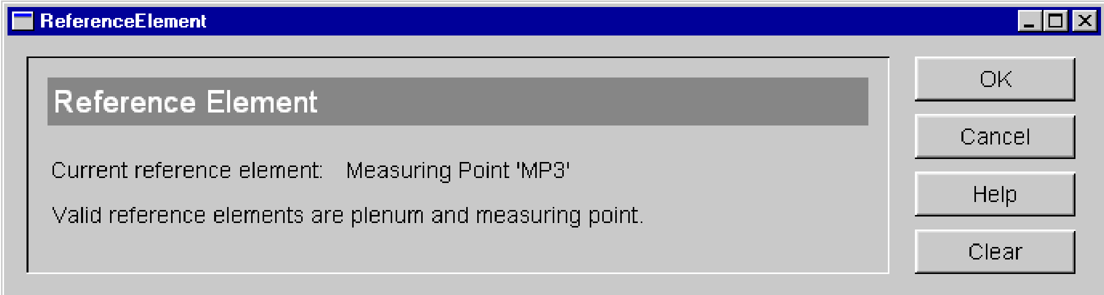


Figure 4—2: Reference Point for Volumetric Efficiency

Note: Save the model before starting calculation.

4.4. 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. Select **View Logfile** to view more detailed information on the simulation run produced by the simulation kernel.

Once the job is complete select **Close** to exit.

4.5. Post-processing

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

Select **Simulation**|**Show Summary** to check 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, then select **Model view** from the submenu to display the model below the tree.

Click on the **Layer** icon and then select **Pressure** from Cylinder 1. Select the curve and in the model tree double click on cylinders 2, 3, 4, 5 and 6 to add the relevant curves. In this example temperature is displayed in Layer_2 with the same procedure.

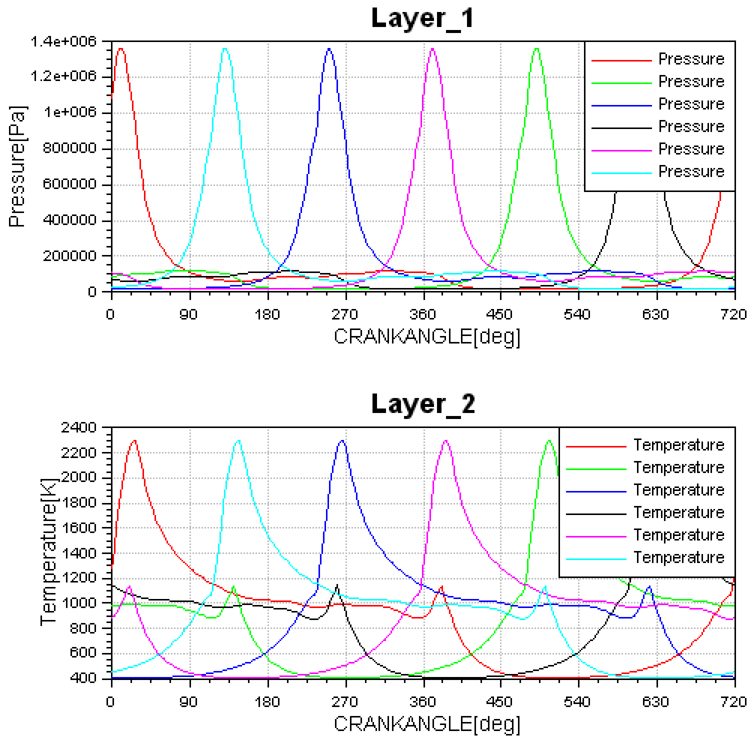


Figure 4—3: Pressure and Temperature Plots From Each Cylinder

5. EXAMPLE FILES

The following example files are included with the **BOOST** program and a short description is given below.

5.1. Acoustic

5.1.1. concentric.bwf

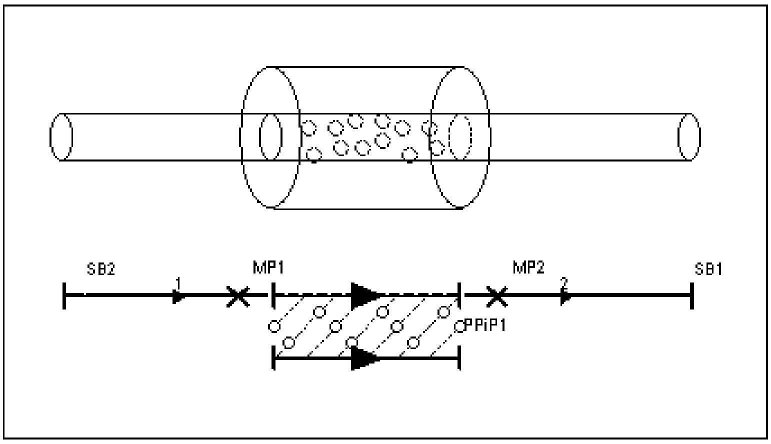


Figure 5—1: BOOST Model of a Concentric Tube Resonator

The **BOOST** model of a concentric tube resonator has been set up to calculate the transmission loss of the component. At the upstream end the boundary type option ‘Acoustic Source’ is used. At the downstream end the boundary type option ‘Anechoic Termination’ is used. Extended measuring points are placed upstream and downstream of the resonator.

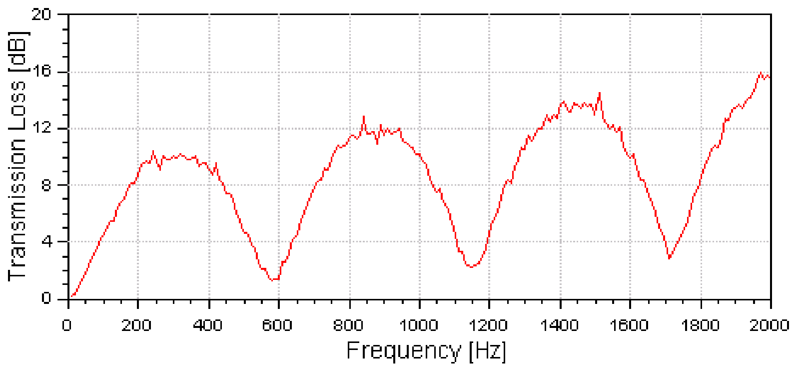


Figure 5—2: Calculated Transmission Loss of the Concentric Tube Resonator

The transmission loss has been calculated from the difference between the calculated linear sound pressure at measuring points 1 & 2 (upstream and downstream of the resonator). The calculated linear sound pressure can be found in the acoustic results folder for all extended measuring points in a model.

5.1.2. rockdrill.bwf

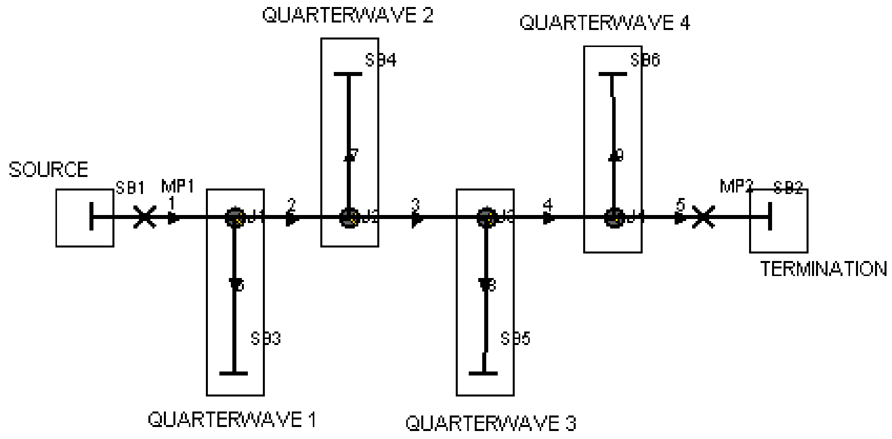


Figure 5—3: BOOST Model of a Rock Drill Silencer

The **BOOST** model is based on an acoustically equivalent system of a rock drill silencer. The setup is similar to the concentric tube resonator case. That is an acoustic source, anechoic termination and two extended measuring points are used.

5.1.3. truck_muffler.bwf

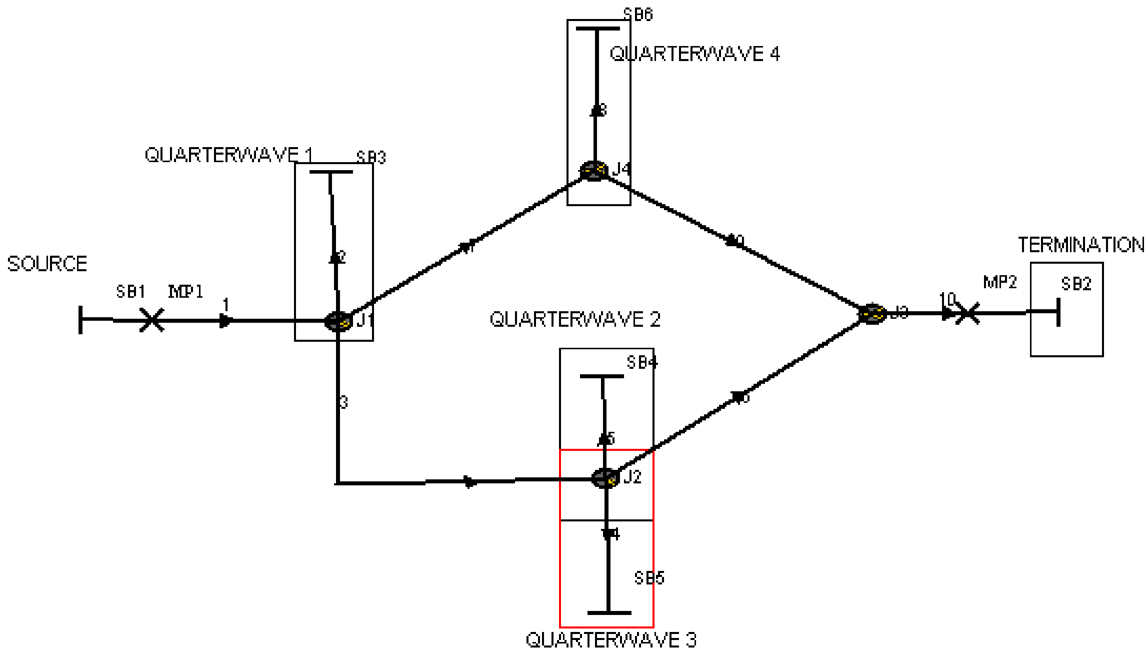


Figure 5—4: BOOST Model of a Truck Muffler

The **BOOST** model is based on an acoustically equivalent system of a truck muffler. The setup is similar to the concentric tube resonator case, i.e. an acoustic source, anechoic termination and two extended measuring points are used.

5.2. Aftertreatment Analysis

In this section, example for the application of the **BOOST** aftertreatment analysis mode are summarized. Additionally it is shown how aftertreatment analysis models are combined with models of **BOOST** cycle simulations. For more detailed information about simulation of aftertreatment elements refer to the **BOOST** Aftertreatment Manual and the **BOOST** Users Guide.

5.2.1. Heatup_1Cat_2D.bwf

This example shows a catalytic converter simulations in aftertreatment analysis mode. It describes the heating up of a catalyst taking into account radial heat loss in a 2D simulation. A catalytic converter with an initial temperature of 373K is flown through with hot gas of 623K. All the conversion and storage reactions are switched off (**KINETIC MODEL NONE, STORAGE MODEL NONE**). The results (see Figure 5—5) show the transient behavior the converter, how it heats up, and how radial heat loss slow down the heating up especially near the converter border.

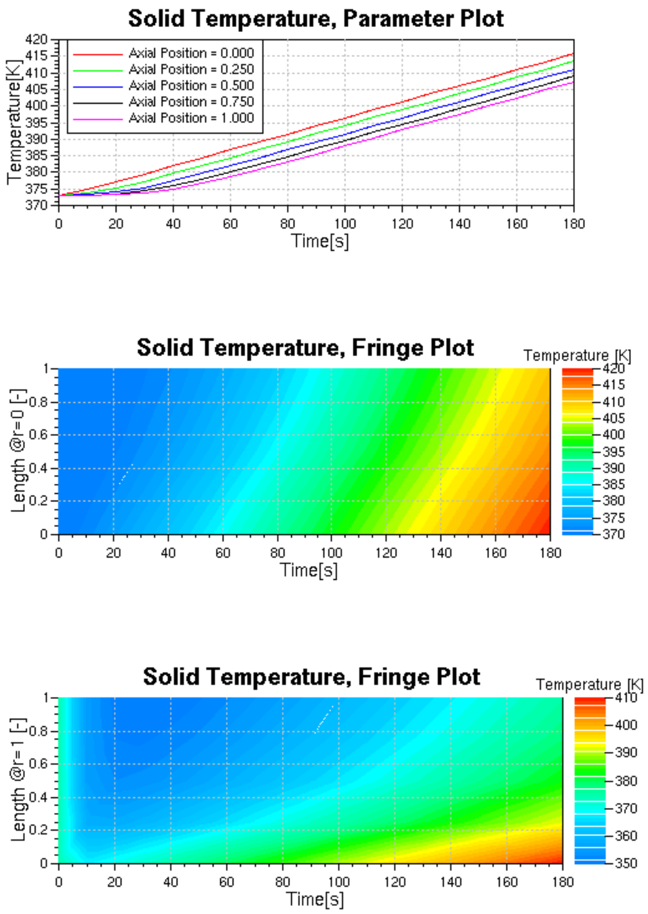


Figure 5—5: Catalyst Heat-up, Solid Temperature as Function of Time and Position

5.2.2. Cat_PressureDrop_Diff_Re_Approaches.bwf

This example shows the influence of the Reynolds friction parameters on the overall converter pressure drop. Three different simulation cases are considered (see Case Explorer), where the parameters for the evaluation of the laminar friction coefficient are varied. As shown in Figure 5—6 different shapes for the pressure drop as function of the velocity are calculated. For the case of a=64, b=-1 (i..e Hagen-Poisseuille law) a linear dependency can be observed.

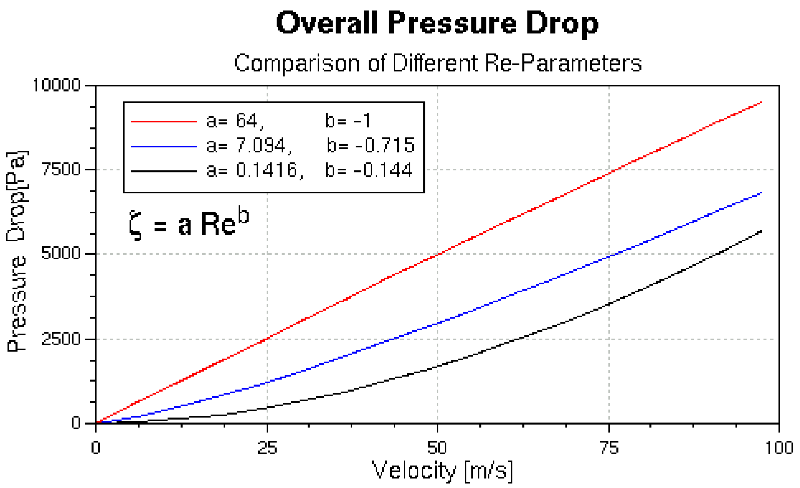


Figure 5—6: Comparison of the Pressure Drop for Different Re-Parameters

5.2.3. DOC_LightOff.bwf

This example shows the light-off behavior (see Figure 5—7) of a Diesel Oxidation Catalyst. The entire example follows the setup of the example `OxiCat_LightOff.bwf` in section 5.2.4. The difference to this example is that here the dedicated pre-defined DOC reaction mechanism is applied.

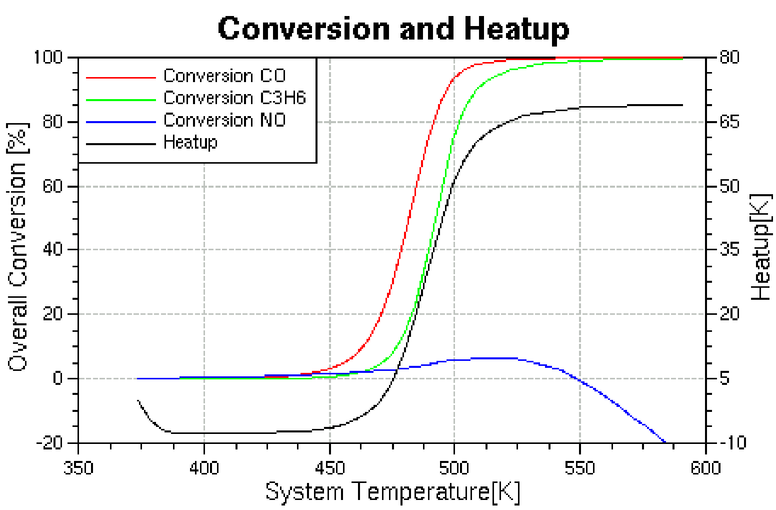


Figure 5—7: DOC Light-Off, Conversion of CO, C3H6, NO and Converter Heatup

5.2.4. OxiCat_LightOff.bwf

This example shows the light-off behavior (see Figure 5—5) of a catalytic converter. A detailed description of the entire example and a step-by-step demonstration of its setup is available in the **BOOST** Aftertreatment Primer Manual.

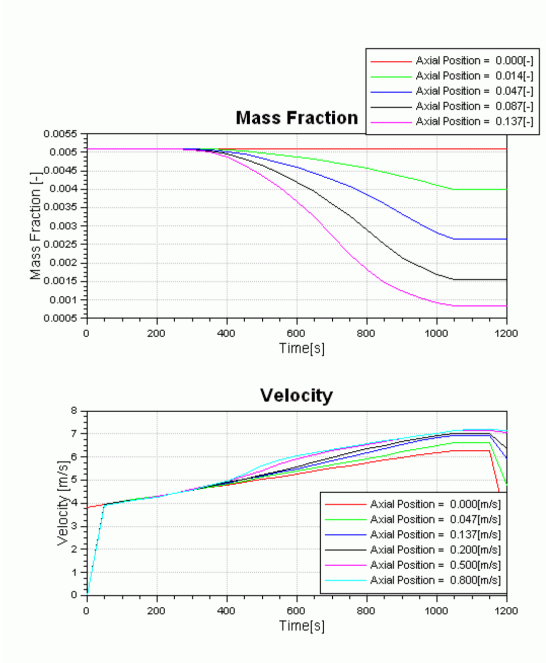


Figure 5—8: Catalytic Converter Light-Off, Solid Temperature and CO-Mass Fraction

5.2.5. HC_Trap_LightOff.bwf

This example shows the light-off behavior (see Figure 5—9) of a catalytic converter with *HC-Trap functionality*. HC (green curve, represented by C3H6) shows conversion at low temperatures because it is adsorbed in the catalyst and not because it is oxidized. The negative conversion of HC indicates a temperature range where HC is desorbed from the surface (more HC is leaving the catalyst than entering). After this desorption phase a step increase of the HC curve represents the real light-off of this species. The light-off curves of CO and NO show a typical behavior. The peak in the NO-curve (blue) is caused by the heat-release during the HC light-off.

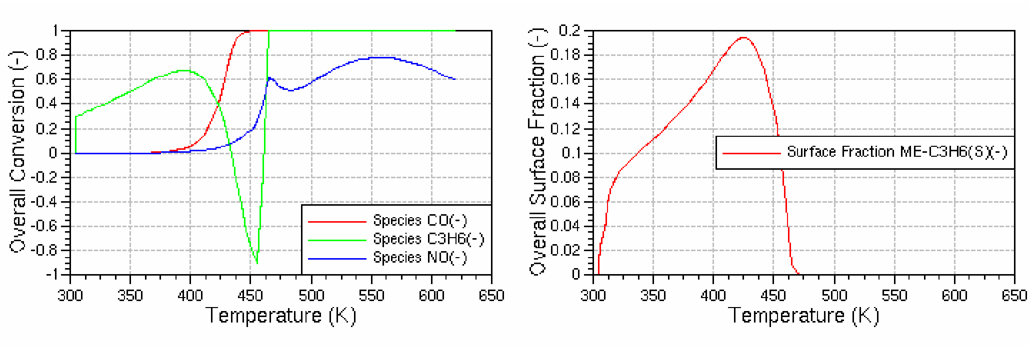


Figure 5—9: HC-Trap Light-Off, Species Conversion and Surface Coverage Fraction of HC

5.2.6. OxiCat_ECE-Cycle.bwf

This example shows the simulation of a catalytic converter during the first 1200 seconds of a drive cycle test. Therefore inlet conditions (mass flux, temperature and composition) are given as transient data via input tables (e.g. see Figure 5—10).

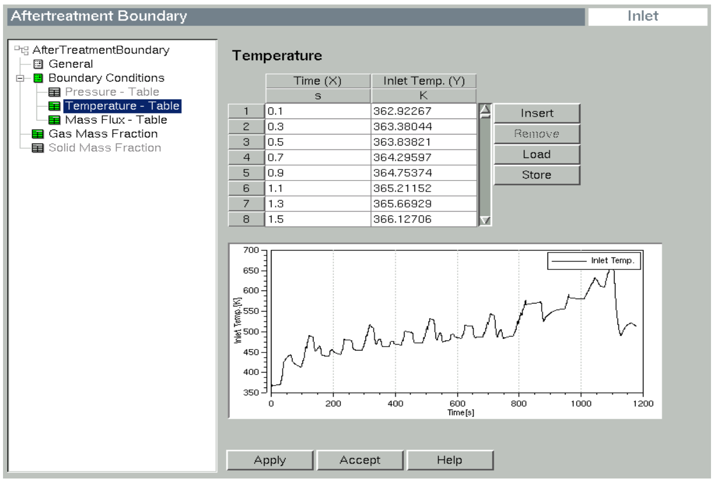


Figure 5—10: Transient Inlet Temperature of a Drive Cycle Test

The results of this drive cycle simulation show how the entire converter heats up and at which times the reactions start and the outlet mass fractions of the major pollutants decrease.

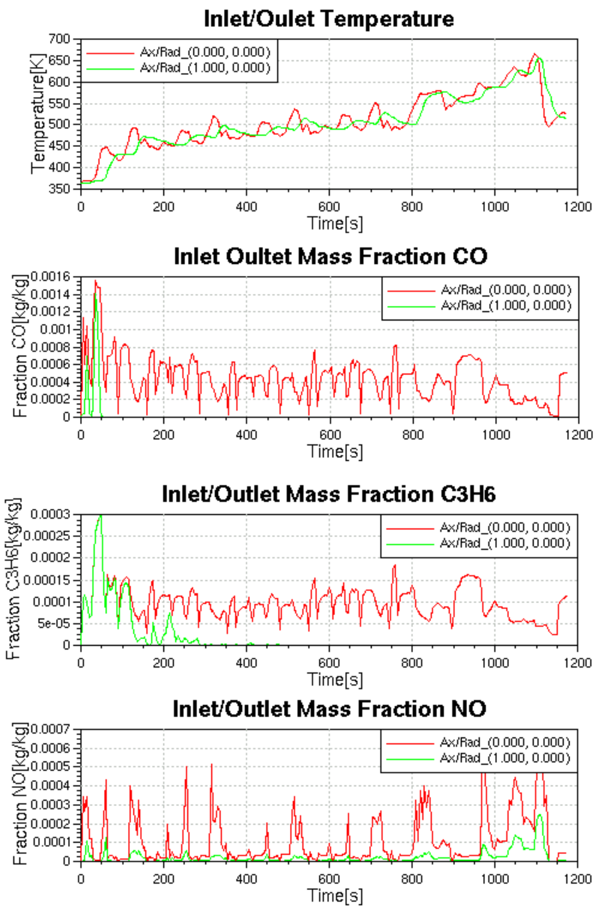


Figure 5—11: Inlet/Outlet Temperature and Mass Fractions During a Drive Cycle

5.2.7. UserRate_Cat_LH_and_Storage.bwf

This example shows the light-off behavior (see Figure 5—12) of C3H6 combined the effect of O2 storage on cerium. The entire reaction model is specified in the routine `mod_userdef_cat.f90`. The result of O2 conversion in the example is caused by its reaction with C3H6 but also by the transient effect O2 storage.

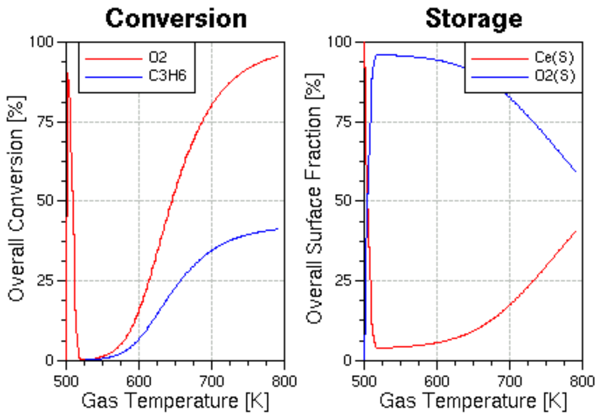


Figure 5—12: User-Rate Light-Off, Conversion of C3H6, O2 and Surface Site Fractions

5.2.8. SCRCat_LightOff.bwf

This example shows the steady-state pre-defined reaction model for SCR converters. All reaction rates are formulated in a steady-state way, either by Eley-Rideal or power-law mechanisms. For the input of all required model parameters the functionality of the "User Defined Parameter" window is used. Two different cases are simulated for different inlet gas compositions. One simulation assumes only the presence of NO (no NO2) and the other simulations assumes an equimolar inlet composition of NO to NO2. For both cases a light-off is calculated. Figure 5—13 shows the conversion of NO, NO2 and NH3 as function of the system temperature. It can be seen that in the presence of NO2 the conversion is higher especially at low temperatures. More detailed information about the entire model is given in SAE-2005-01-0948.

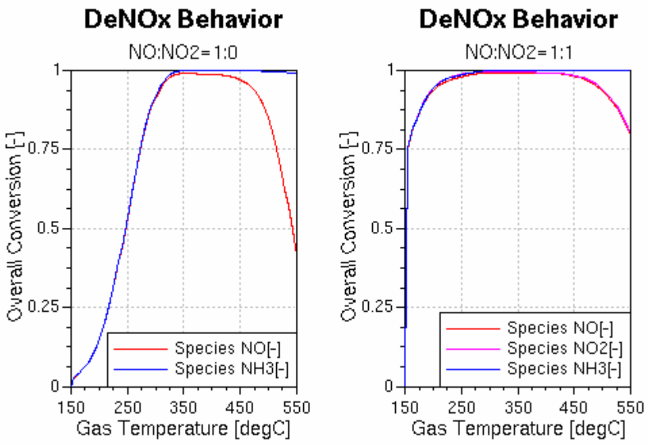


Figure 5—13: SCR Catalyst – Conversion of NO and NO2 for Different Inlet Gas Compositions

5.2.9. SCRCat_HSO_LightOff.bwf

This example shows the pre-defined reaction model for HSO converter. For the input of all required model parameters the functionality of the "User Defined Parameter" window is used. In the model three different reaction sections are assumed. In the first section (H) the hydrolysis of HCNO to NH3 and CO2 is modeled. In the second section (S) SCR reactions for the conversion of NO and NO2 with NH3 are taken into account. In the third section (O) an accelerated reaction for the oxidation of NH3 is assumed caused by a different coating compared to the two first sections. Figure 5—14 shows how the mole fraction of NH3 changes over the length of the converter. More detailed information about the entire model can be found in SAE-2005-01-0948.

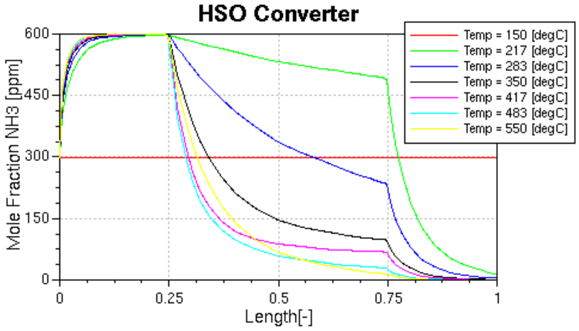


Figure 5—14: HSO-Converter – Spatial NH3 Distribution at Different System Temperatures

5.2.10. SCRCat_Trans_AdDesorption.bwf

This example shows the transient pre-defined reaction model for SCR conversion. The transient behavior of the NH3 ad/desorption at the surface is explicitly taken into account by solving surface site balance equation. For the input of all required model parameters the functionality of the "User Defined Parameter" window is used. Two different cases are simulated for different inlet gas compositions.

First, the ad/desorption of NH3 is simulated in the absence of NO. Second the NO conversion is calculated. Figure 5—15 shows for both cases the inlet and outlet mole fraction of NH3. In the case without NO (left diagram) a delay of the outlet fraction compared to the inlet values can be observed. This is caused by the storage of and release of NH3 at/from the surface. In the presence of NO, a lowered outlet fraction of NH3 is given caused by its consumption in the SCR reaction. More detailed information about the entire simulation example can be found in SAE-2005-01-0948.

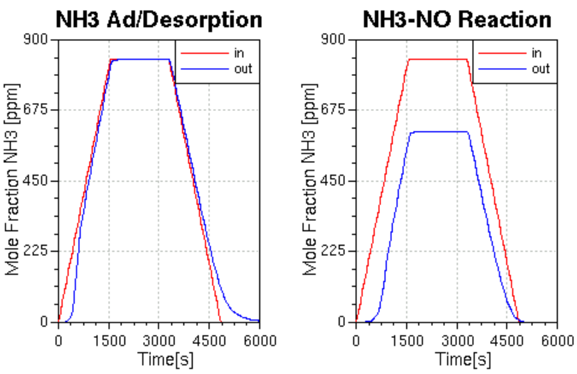


Figure 5—15: SCR Catalyst – Transient Behavior of NH3 Ad/Desorption with and without NO

5.2.11. DPF_Loading.bwf

This example shows the simulation of a Diesel Particle Filter during loading. All regeneration reactions were switched of (**Regeneration Mode None**) and the filter was supplied with a soot flux at its inlet boundary condition. As shown in Figure 5—16, the soot mass in the filter increases over time and therefore also the pressure drop of the DPF goes up. The effect of soot migration can be seen at the length coordinate zero where no soot loading is given. The influence of the solver tolerance is demonstrated using the functionality of the “User Defined Parameters”. Therefore two simulation cases are specified in the Case Explorer with different levels of refinement. The Parameter Key needed in the list of “User Defined Parameters” is “ATM_DPF_REFINE_FLOW_SOLUTION”.

If the user specifies a value of 1, the default solver settings are applied. If values greater than one (i.e. 100) are specified the tolerance is refined by this factor. Higher solver tolerances increase the CPU time but decrease numerical noise (see Figure 5—17) in the velocity and pressure results.

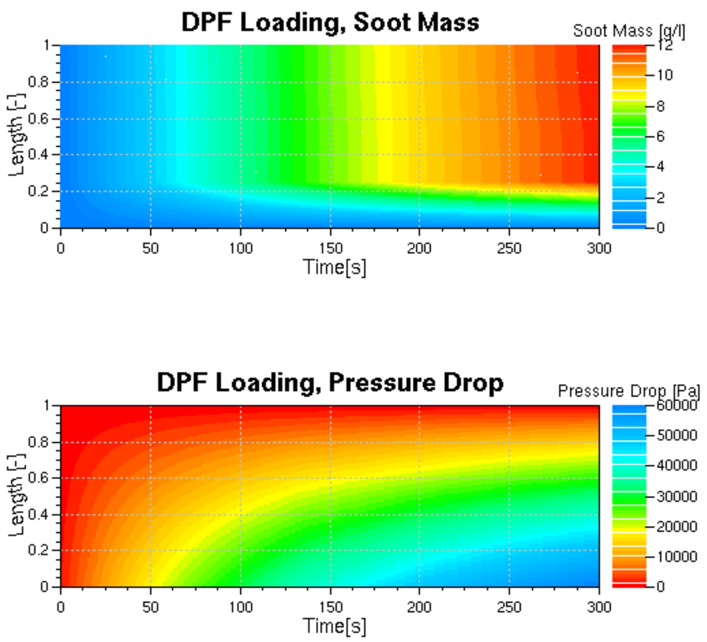


Figure 5—16: DPF Loading - Soot Mass and Pressure Drop as Function of Time and Filter Length

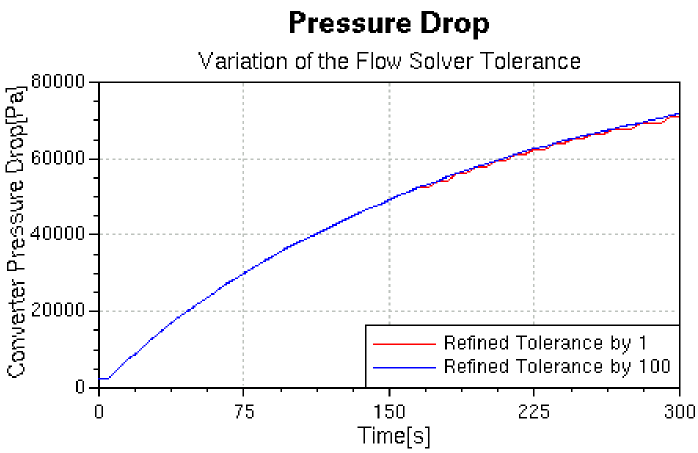


Figure 5—17: DPF Loading – Pressure Drop for Different Solver Tolerances

5.2.12. DPF_BareTrap_Regeneration.bwf

This example shows the simulation of a Diesel Particulate Filer (DPF) regeneration. The model is supplied with two different boundary conditions for the inlet mass flow. The mass flow is stored in two input files `DPF_BareTrap_massflux_bnd_1.dat` and `DPF_BareTrap_massflux_bnd_2.dat` that are automatically reloaded for the individual simulation cases (see also Case Explorer). A detailed description of this example (see Figure 5—18) is available in the **BOOST** Aftertreatment Primer Manual.

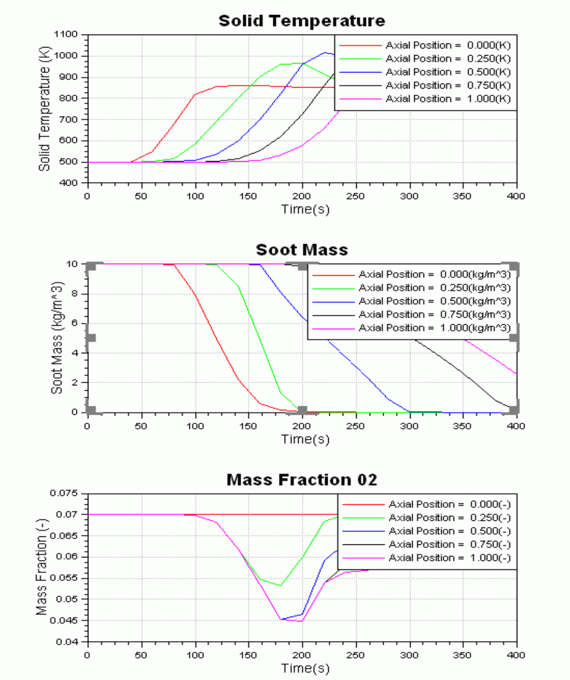


Figure 5—18: DPF Regeneration, Temperatures and Species Mass Fractions

5.2.13. DPF_BareTrap_and_CSF_Regeneration_CSF.bwf UserRate_DPF_BareTrap_and_CSF_Regeneration.bwf

These two examples show how catalytic reactions are additionally taken into account during a DPF regeneration. Both cases assume that the filter wall is catalytically active over 50% of the entire wall thickness. Within this catalytic wall section the conversion CO and C3H6 is modeled with rate equations of Langmuir-Hinshelwood type. Figure 5—19 summarizes the transient behavior of the filter temperature and soot mass during the regeneration. A comparison to Figure 5—18 shows the impact of the catalytic reactions. The combustion of CO and C3H6 in the filter-wall additionally produces heat that shifts the initiation of the soot combustion to lower temperatures. The reaction constants for the catalytic reactions were assumed in order to show the effect of these reactions in a principle way. For more detailed simulations these parameters need to be adjusted with the help of experimental data. The example case `DPF_BareTrap_and_CSF_Regeneration.bwf` shows how the CO/C3H6 conversion is modeled via a pre-defined reaction model. The example case UserRate_`DPF_BareTrap_and_CSF_Regeneration.bwf` shows how the same reactions are modeled via a user-routine. This routine can be accessed by the user and therefore also modified.

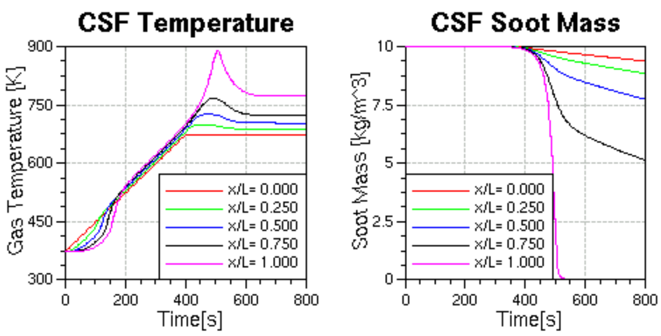


Figure 5—19: Catalytic Soot Filter - Temperature and Soot Mass During Regeneration

5.2.14. DPF_BareTrap_Regeneration_Variab_cp.bwf

This example shows how a temperature dependent heat capacity of the DPF filter substrate can be modeled. Figure 5—20 shows the impact of different heat capacities. In the first case (red lines) it is assumed that the heat capacity changes between 1250J/kgK and 1800J/kgK. In the second case (blue lines) it is kept constant at a value of 1250J/kg/K. The results show lower maximum temperatures and slower soot conversion at higher heat capacities.

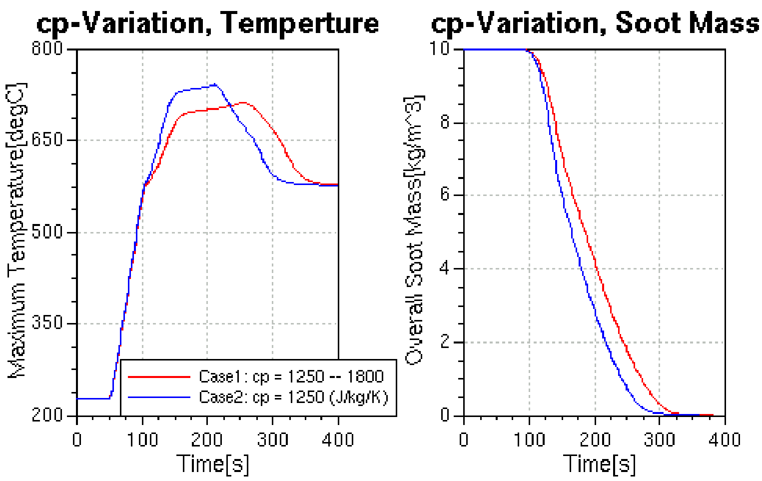


Figure 5—20: Variable DPF Heat Capacity - Temperature and Soot Mass During Regeneration

5.2.15. DPF_BareTrap_Regeneration_Variab_Permeab__UP.bwf

This example shows how a temperature dependent soot permeability can be modeled. Therefore the functionality of the "User Defined Parameter" window is used. It is assumed that the soot permeability changes linearly with temperature. For two temperatures, 300K and 1000K, two input values need to be specified by the user. First, in the "Soot and Filter Properties" window the input value "Soot Permeability" refers to the temperature of 300K. Second, in the "User Defined Parameter" window the key "ATM_SOOT_PERM_1000K" specifies a permeability at 1000K. At temperatures between these two borders (300K, 1000K) the permeability is linearly interpolated. At temperatures outside this range the border value is kept constant. Figure 5—21 shows the impact of different soot permeabilities. In the first case (red lines) it is kept constant at a value of 1e-13m^2. In the second case (blue lines) it changes between 1e-13m^2 and 5-13m^2. The results show a lower pressure drop with higher soot permeability and a significantly changed maximum temperature profile.

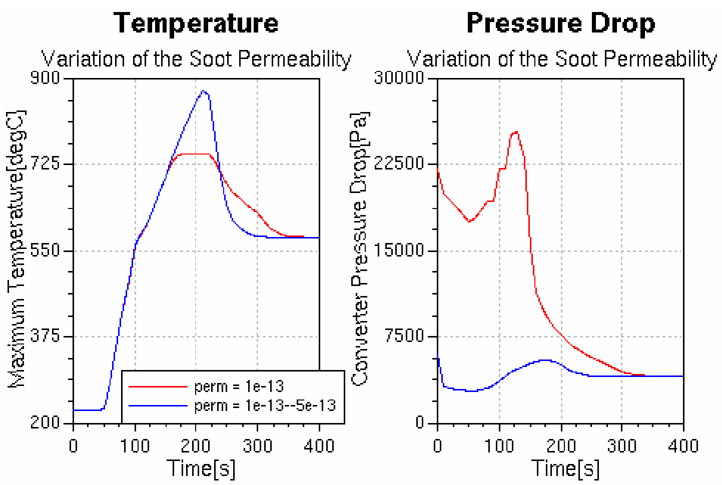


Figure 5—21: Variable Soot Permeability - Temperature and Pressure Drop During Regeneration

5.2.16. DPF_Control_BareTrapRegen.bwf

This example shows how the *Formula-Interpreter* element can be used to control the maximum temperature during a forced regeneration event. Therefore the example DPF_BareTrap_Regeneration.bwf (see section 5.2.12) is extended with the Formula-Interpreter element. This element sensors maximum temperature in the DPF after each time step and changes the inlet mass flow and inlet O2/N2 mass fractions if a given maximum DPF temperature is exceeded. Figure 5—22 shows a comparison of the maximum filter temperatures for a controlled and uncontrolled regeneration.

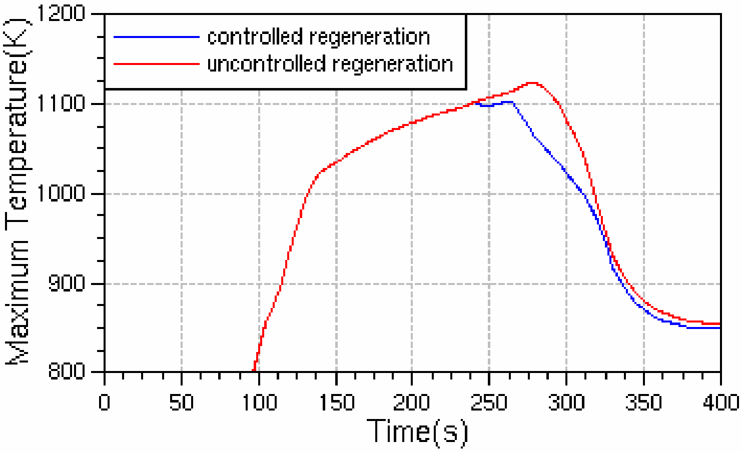


Figure 5—22: DPF Control – Maximum Temperatures During controlled uncontrolled Regeneration

5.2.17. DOC1_an_SCR1.bwf

This example shows how a system of one pre-oxidation catalyst and one SCR converter is simulated. In the DOC only the oxidation of NO is enabled for reasons of simplicity. In the SCR converter the pre-defined steady SCR reaction model is chosen. The entire system is heated up from 423K to 873K and the NOx conversion of the SCR, as final result, is calculated (see Figure 5—23).

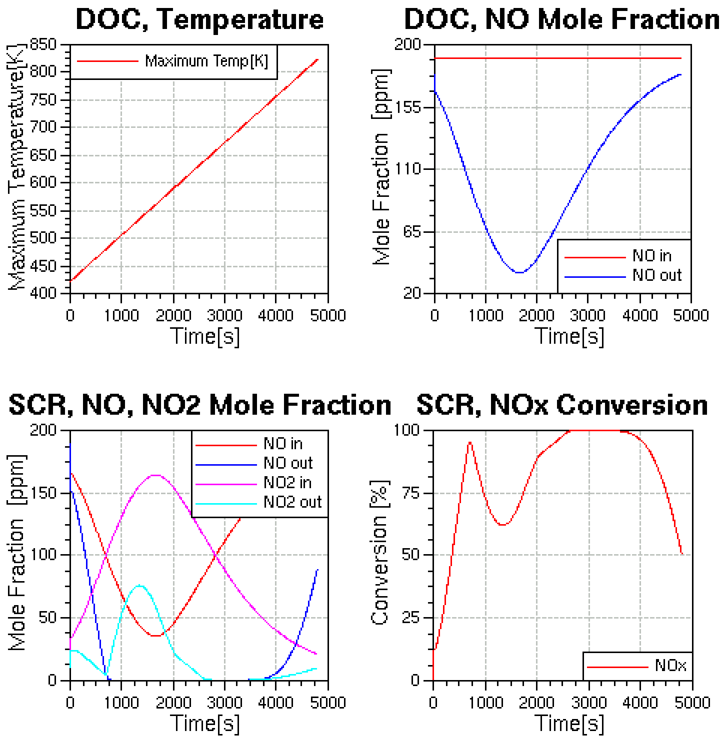


Figure 5—23: DOC and SCR – Temperatures Mole Fractions of NO/NO2 and NOx Conversion

5.2.18. DOC1_an_DPF1.bwf

This example shows how a system of one pre-oxidation catalyst and one DPF is simulated. The configuration of the DOC model is setup following the example `OxiCat_LightOff.bwf` and the DPF model is specified following the example `DPF_BareTrap_Regeneration.bwf`. The heatup of the catalyst is generated by increasing the inlet mass fractions of CO and HC. Based on the increased inlet temperatures the forced regeneration in the DPF is initiated and the soot in the filter is completely combusted (see Figure 2—6).

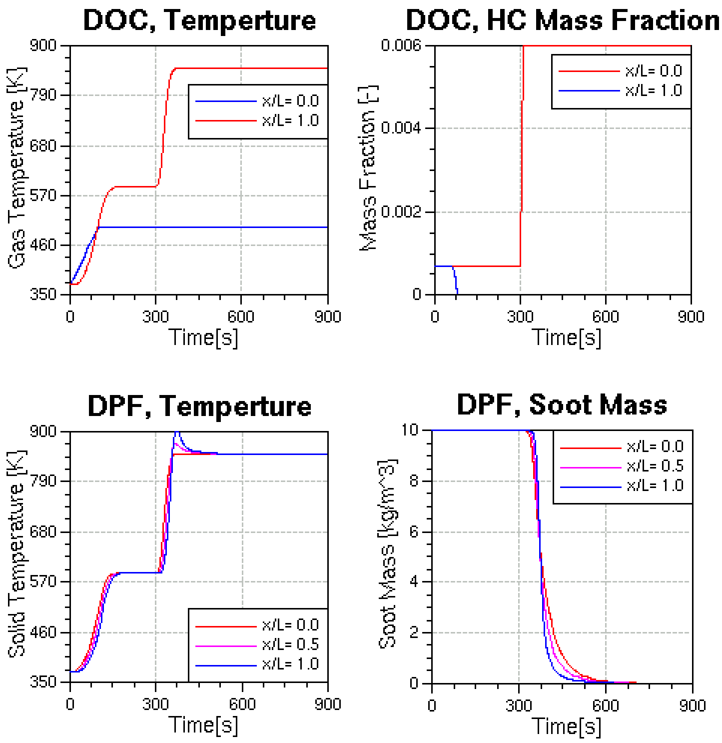


Figure 5—24: DOC and DPF – Temperatures, HC Mass Fractions and Soot Mass

5.2.19. DPF_BareTrap_Regeneration_with_Ash_and_Plugs.bwf

This example shows how DPF bare trap regeneration is simulated in the presence of ash plugs of different plug lengths. The results (see Figure 5—25) show for the given example no pronounced difference for the maximum temperature curves. More pronounced differences are given for the simulated transient pressure curves.

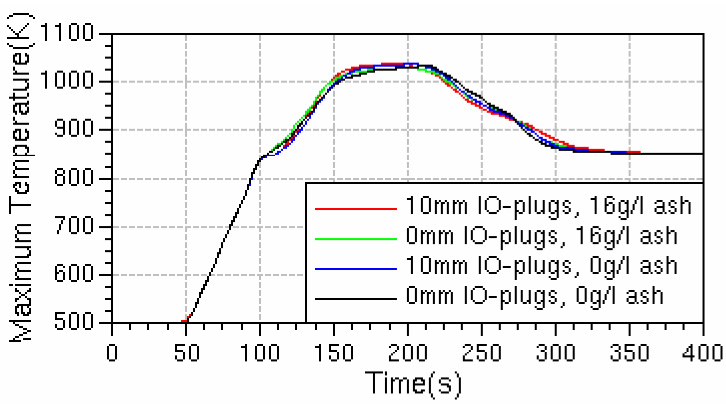


Figure 5—25: DPF Regeneration with Ash-Plugs – Maximum Temperatures

5.2.20. SCRCat_TransKinetics__GE_and_ATM.bwf

This example shows how the transient SCR reaction scheme (see example described in Section 5.2.10) is applied within the *BOOST Gas Exchange* simulation task and within the *BOOST Aftertreatment Analysis* simulation task. Both simulation tasks use completely different numerical approaches to solve the gas dynamics within the catalyst. Since both solvers (Gas Exchange and Aftertreatment) apply the same reaction mechanism similar results for similar boundary conditions are expected. This is shown in Figure 5—26.

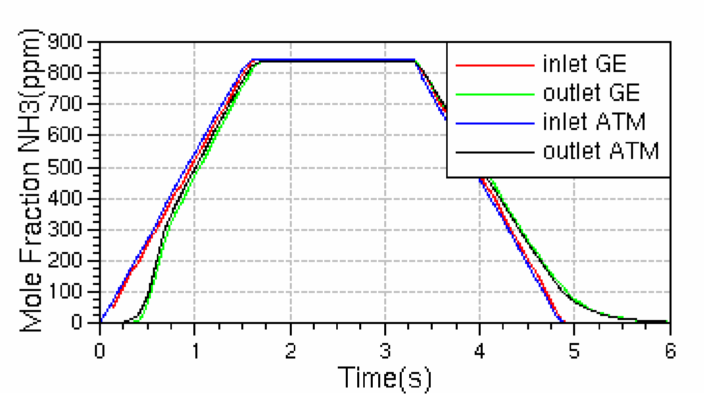


Figure 5—26: NH3 Storage Simulation – Comparaison Aftertreatment Mode vs. Gas Exchange Mode

5.2.21. DOC_LightOff__GE_and_ATM.bwf

This example shows how the light-off of a DOC (see example described in Section 5.2.3) is simulated within the *BOOST Gas Exchange* simulation task and within the *BOOST Aftertreatment Analysis* simulation task. Both simulation tasks use completely different numerical approaches to solve the gas dynamics within the catalyst. Since both solvers (Gas Exchange and Aftertreatment) apply the same reaction mechanism similar results for similar boundary conditions are expected. This is shown in Figure 5—27.

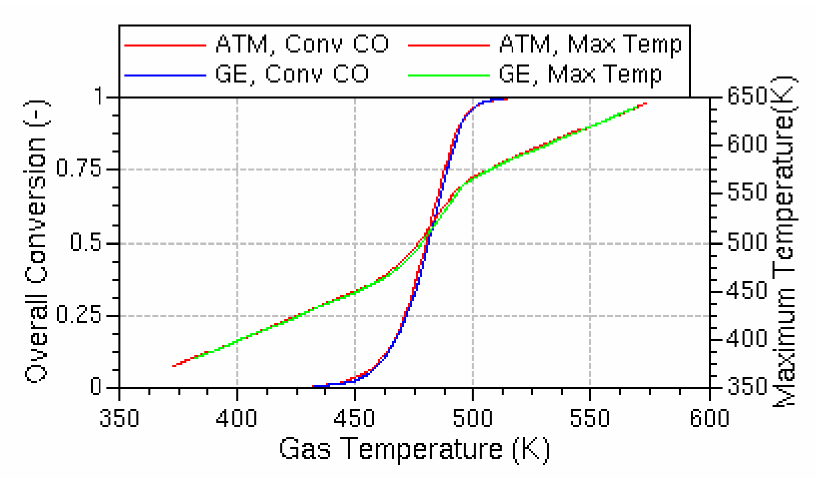


Figure 5—27: DOC Light-Off Simulation – Comparison Aftertreatment Mode vs. Gas Exchange Mode

5.2.22. TWC_LightOff_and_LambdaSweep.bwf

This examples show the light-off simulation of a TWC where the gas composition at the inlet boundary changes from lean to rich conditions. The results summarized in Figure 5—28 show the conversion of CO and NO as function of time (correlates with linearly increasing inlet temperature) for different inlet redox-ratios. A redox-ratio smaller than 1 indicates lean conditions, 1 represents stoichiometric conditions and values higher than 1 indicate rich conditions.

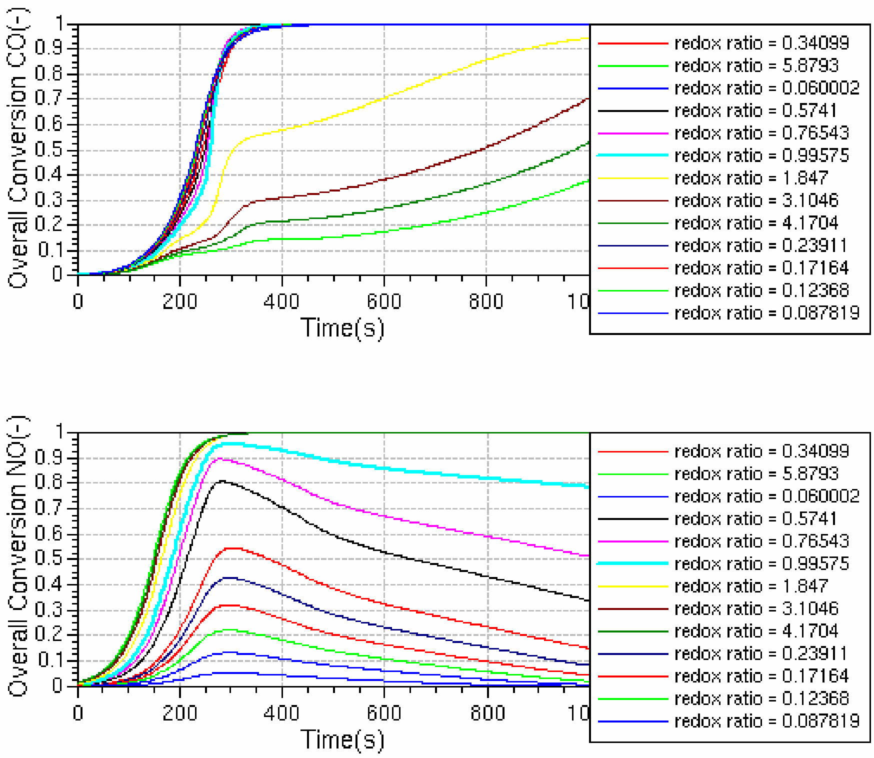


Figure 5—28: TWC-Light-Off Simulation – CO and NO Conversion for Different inlet Redox-Ratios

5.2.23. DPF_pdrop_vs_FlowVel.bwf

This example shows the impact of the DPF channel diameter ratio on the overall pressure drop of an empty filter. As given in Figure 5—29, the pressure drop shows a stronger increase for filters with octo-square structures compared to a “normal” squared cell filter.

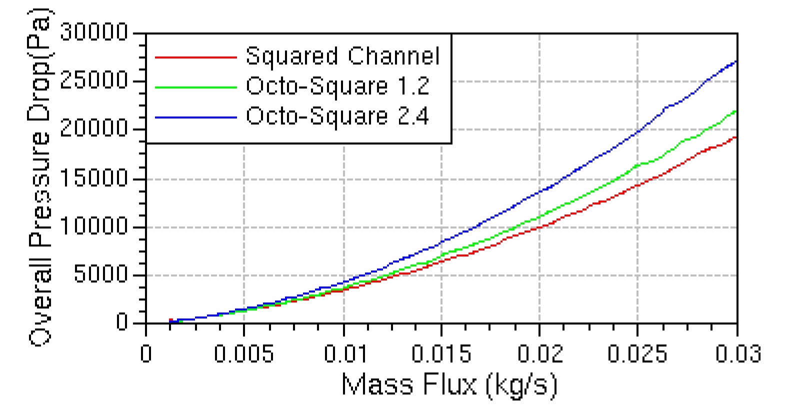


Figure 5—29: DPF Pressure Drop Simulation – Comparison of Unloaded Filters with Different Channel Structures

5.2.24. DPF_pdrop_of_DepthAndCakeLoading.bwf

The example shows the impact of depth and cake filtration on the overall pressure drop during transient loading of the different filter types. Figure 5—30 shows that the overall pressure drop changes its gradient as a function of the soot loading. This indicates the existence of individual depth and cake layer filtration regimes.

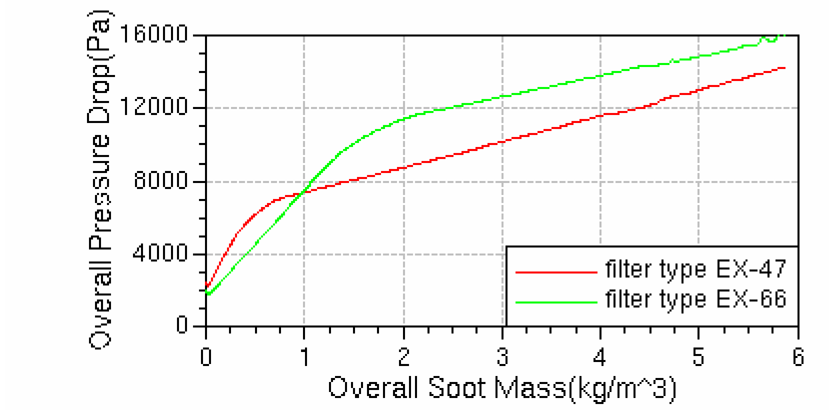


Figure 5—30: DPF Pressure Drop Simulation –-Pressure Drop due to Depth and Cake Filtration for Two Different Filters

5.2.25. DPF_pdrop_vs_AshDistribution.bwf

This example shows the impact of ash and its distribution on the overall pressure drop of a DPF. Figure 5—31 illustrates the overall filter pressure drop as a function of an ash distribution factor for filters with two different ash-loadings and cell structures. An ash distribution factor of 1 means the entire ash is stored as plug at the end of the inlet channel. A distribution factor of 0 means that the entire ash is forming an ash layer.

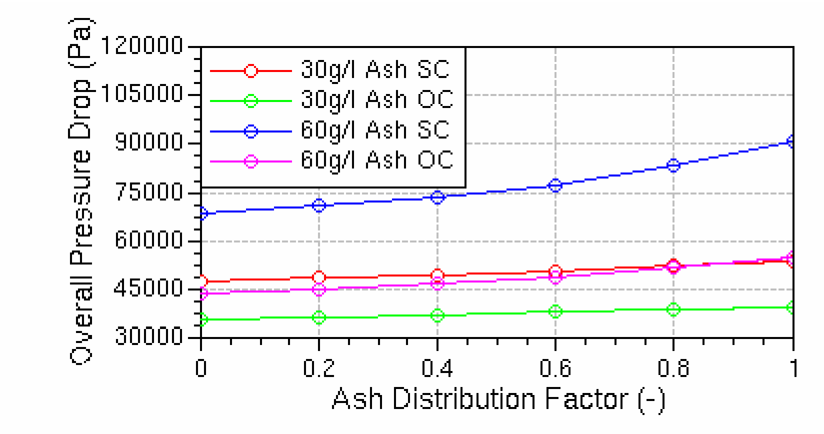


Figure 5—31: DPF Pressure Drop Simulation – Overall Pressure Drop for Different DPF Types (Square Channel, Octosquare) and Different Ash Loadings

5.2.26. DPF_Depth_and_Cake_Loading_Regeneration.bwf

This example shows the transient changes of soot mass located in the depth and cake filtration layer during loading and regeneration. Figure 5—32 illustrates that soot located in both layers gets regenerated. Only the soot cake layer increases during the loading phase. This is because of the existence of an ash layer.

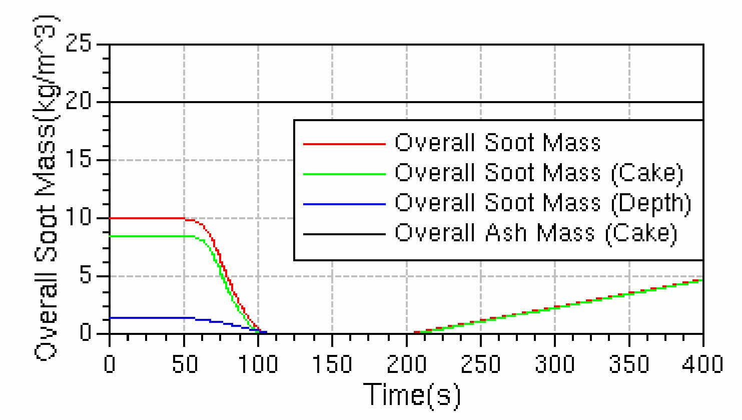


Figure 5—32: DPF Loading and Regeneration Simulation – Overall Soot Loading, Soot Loading in the Depth and Cake Layer and Ash Loading

5.2.27. DPF_CRT_9Steps.bwf

This example shows a CRT regeneration for two different assumptions of the cake filtration layer. The CRT regeneration reactions are assumed to take place within the depth filtration layer of the filter. Depending on the assumption that the cake layer remains immobile or not, different conversion rates can be observed. As shown in Figure 5—33, a faster decrease of the soot mass is simulated if soot from the cake layer slides into the depth filtration layer and therefore also gets converted by the faster CRT reactions.

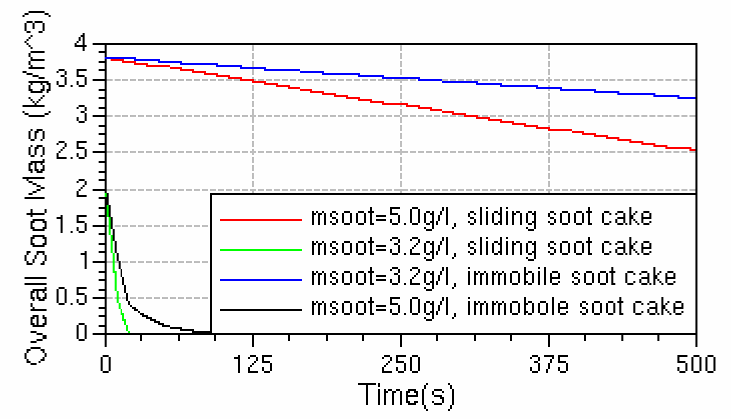


Figure 5—33: DPF CRT Regeneration – Overall Soot Mass Simulated for Sliding and Immobile Soot Cake Layer

5.2.28. Pipe_RadiationImpact_on_HeatLoss.bwf

This example (see Figure 5—34) shows the application of the pipe model in BOOST Aftertreatment. Therefore a dual-wall, air-gap insulated, pipe is considered. The model is calculated twice, once with effect of ration in the air and once without radiation in the air-gap. The comparison of the pipe outlet temperatures (blue curve, calculated with radiation, red curve, calculated without radiation) shows lower temperatures for the first case and therefore the importance of radiation.

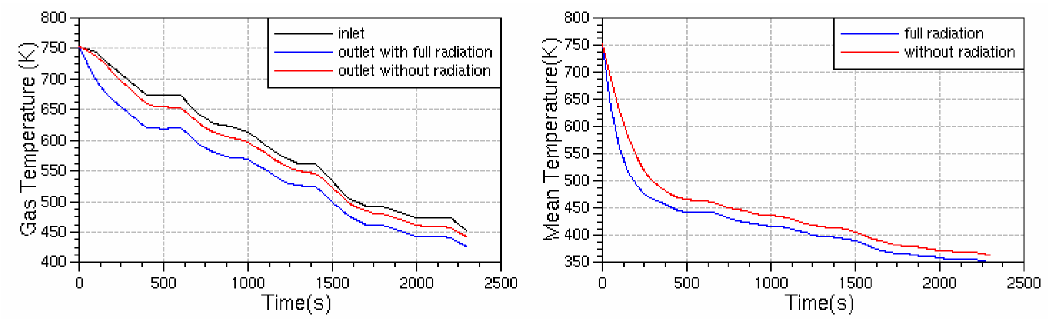


Figure 5—34: Dual-Wall Pipe – Temperatures Considering the Impact of Radiation

5.2.29. WHTC_400s_DOC_DPF_SCR.bwf

This example (see Figure 5—35) shows how BOOST Aftertreatment can be used to simulate an entire exhaust gas system consisting of a DOC, DPF and SCR connected by pipes. The simulation is performed for the first 400s of a WHTC (World Harmonized Transient Cycle). Typical results of this kind of system simulation are the temperatures of the different components and the cumulative emissions of the system.

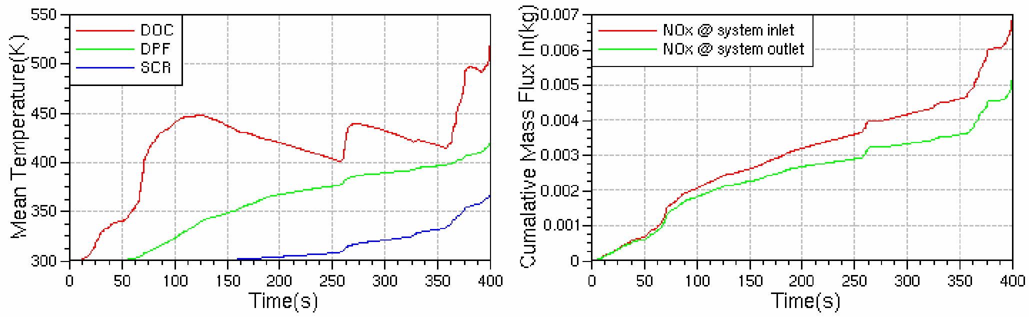


Figure 5—35: System Simulation – Temperatures and Emissions During a Drive Cycle

5.3. Animation

5.3.1. 2t1anim.bwf

Based on the `2t1calc.bwf` example (refer to Figure 5—94) an animation calculation is performed. The following figure displays the snapshot of the pressure animation results in PP3.

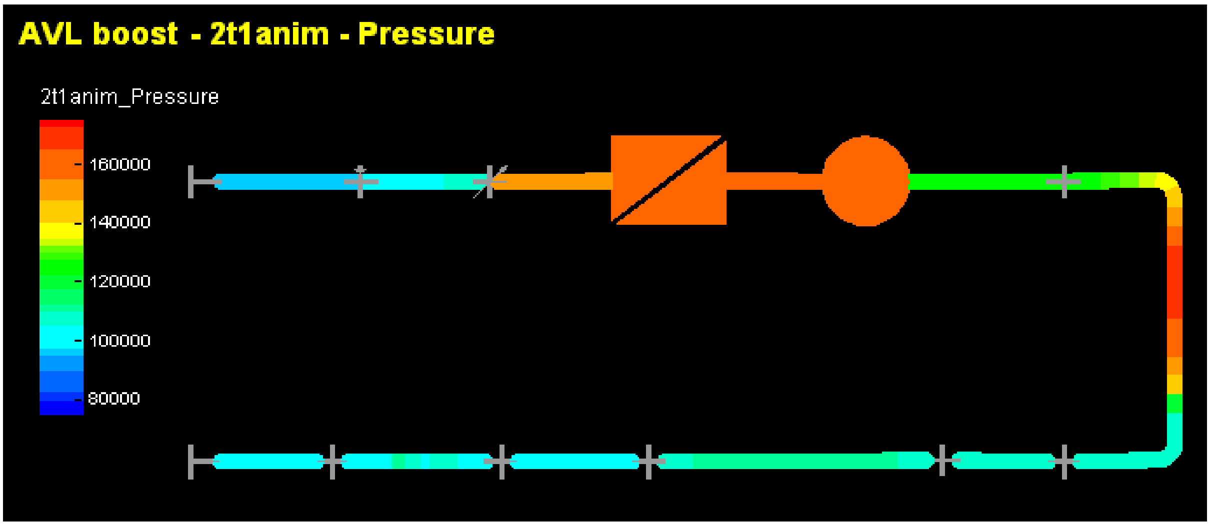


Figure 5—36: Pressure Animation Results for 2t1anim.bwf

5.4. Burn

5.4.1. 4t1burn.brn

The `4t1burn.brn` file contains all data required for performing a combustion analysis. The data is derived from the `4t1calc.bwf` BOOST model of a motorcycle 4-stroke engine.

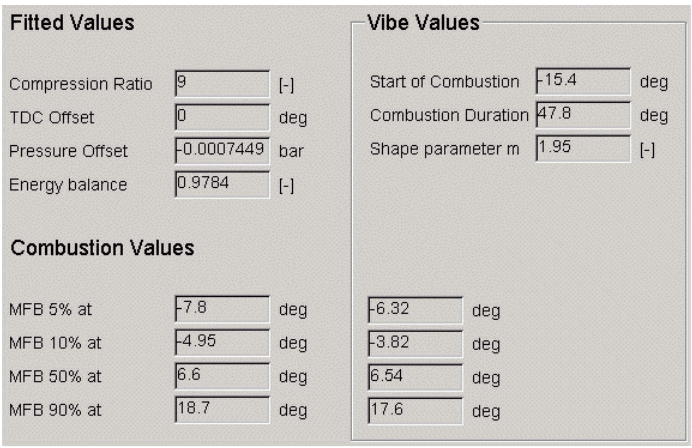


Figure 5—37: Operating Point Results for 4t1burn.brn

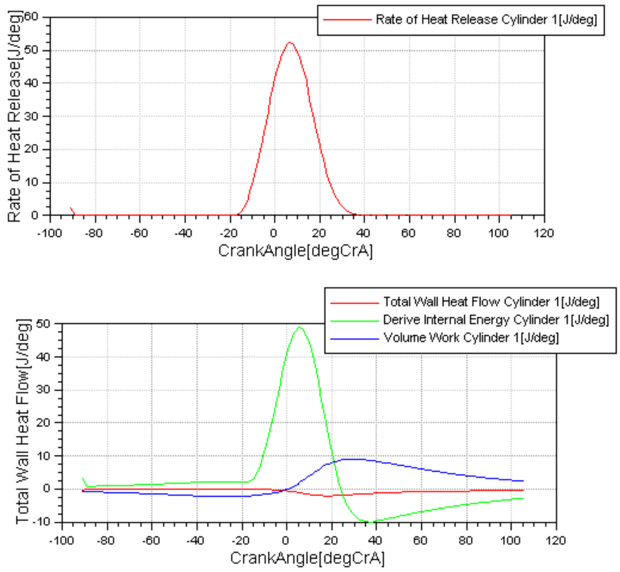


Figure 5—38: Rate of Heat Release Results for 4t1burn.brn

5.4.2. 4t1burn_2zone.brn

The `4t1burn_2zone.brn` file contains all data required for performing a two zone combustion analysis. The data is derived from the `4t1calc.bwf` BOOST model of a motorcycle 4-stroke engine and is identical to 4t1burn.brn except that it uses two zone combustion analysis.

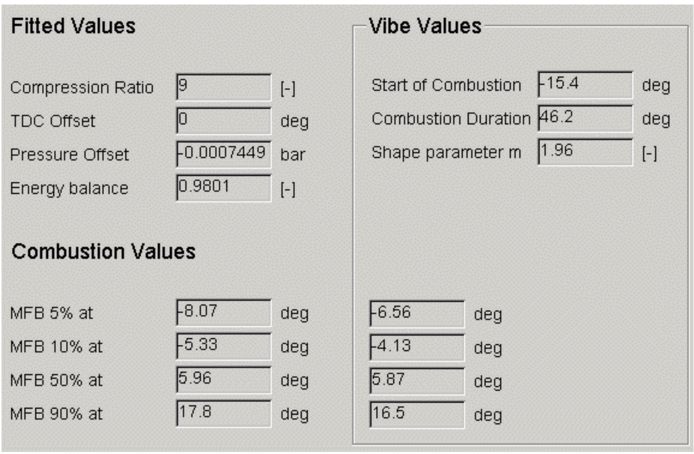


Figure 5—39: Operating Point Results for 4t1burn_2zone.brn

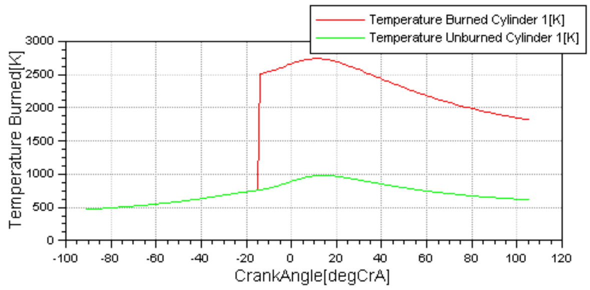


Figure 5—40: Burned and Unburned Zone Temperatures for 4t1burn_2zone.brn

5.4.3. 2t1burn.brn

This is a 2 stroke gasoline example based on the BOOST example `2t1calc.bwf`.

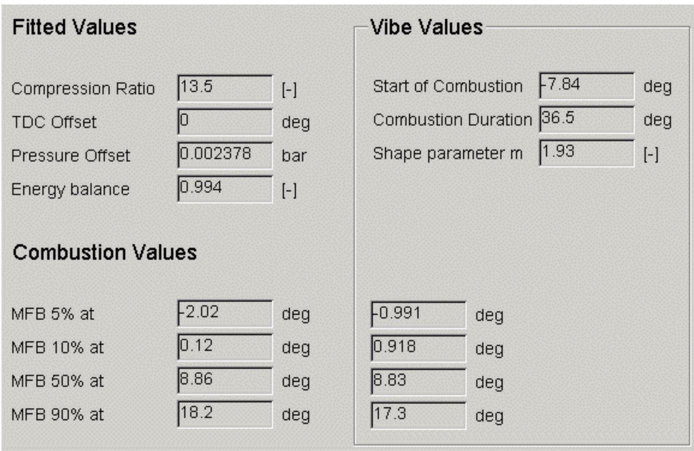


Figure 5—41: Operating Point Results for 2t1burn.brn

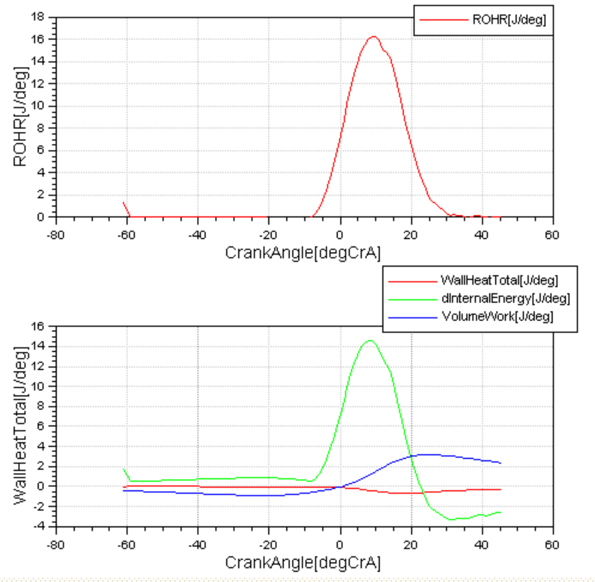


Figure 5—42: Rate of Heat Release Results for 2t1burn.brn

5.4.4. Demo_BurnRate.brn

This is a 4 stroke gasoline example based on the Demo_BurnRate example of GCA (Gas Exchange and Combustion Analysis).

5.4.5. ottoburn.brn

This is a multiple case four stroke gasoline example based on the BOOST example `ottocalc.bwf`. There are a total of four cases:

- 7000 rpm
- 5000 rpm
- 3000 rpm
- 1000 rpm

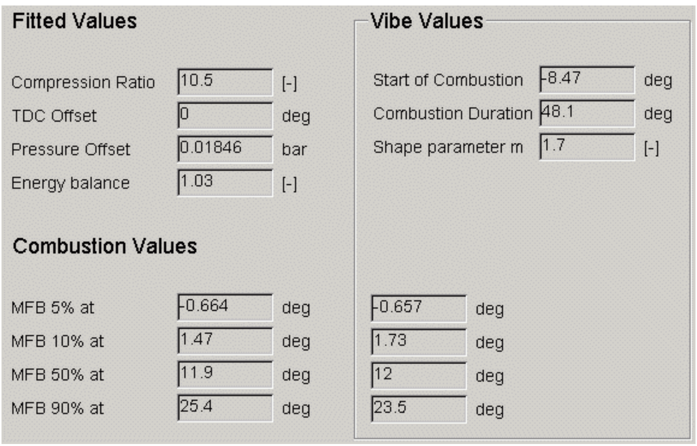


Figure 5—43: Operating Point Results for 7000 rpm case of ottoburn.brn

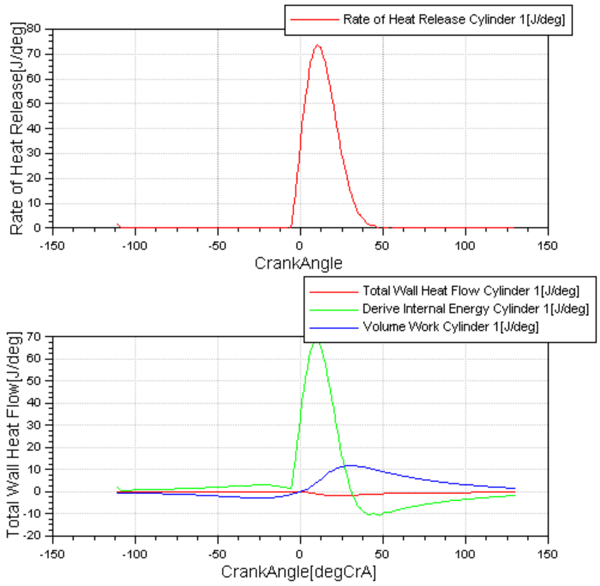
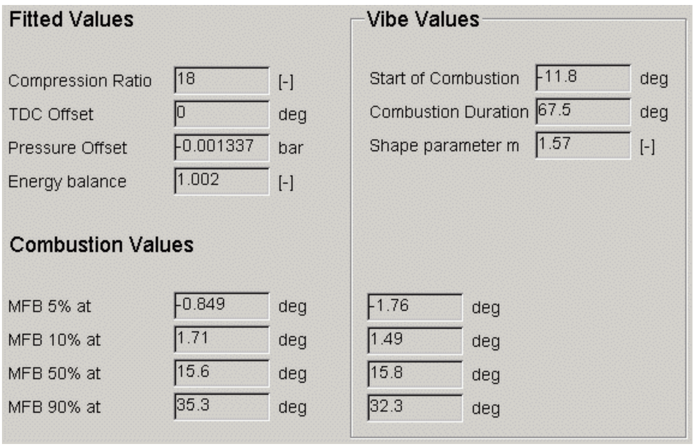


Figure 5—44: Rate of Heat Release Results for 7000 rpm case of ottoburn.brn

5.4.6. tciburn.brn

This is a four stroke diesel testcase based on the BOOST example `tcicalc.bwf`.



Fitted Values

Compression Ratio	18	[-]
TDC Offset	0	deg
Pressure Offset	-0.001337	bar
Energy balance	1.002	[-]

Vibe Values

Start of Combustion	-11.8	deg
Combustion Duration	67.5	deg
Shape parameter m	1.57	[-]

Combustion Values

MFB 5% at	-0.849	deg	-1.76	deg
MFB 10% at	1.71	deg	1.49	deg
MFB 50% at	15.6	deg	15.8	deg
MFB 90% at	35.3	deg	32.3	deg

Figure 5—45: Operating Point Results for tciburn.brn

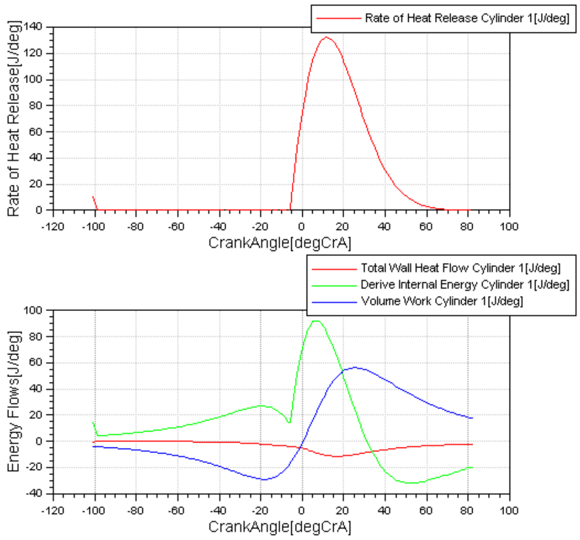


Figure 5—46: Rate of Heat Release Results for tciburn.brn

5.5. Combustion Models

5.5.1. 4t1fractal.bwf

This model is a motorcycle 4-stroke engine. The ROHR is calculated with the Fractal combustion model for a spherical cylinder head and a flat piston crown. It is a 2 valve engine so the spark plug has an offset from the cylinder center.

Based on the `4t1calc.bwf` example (refer to Figure 5—95), a Fractal Combustion Model is used.

In the **Cylinder | Properties | Combustion** window select **Fractal** from the **Heat Release** pull-down menu. Then select **Fractal** in the tree to access the following input fields:

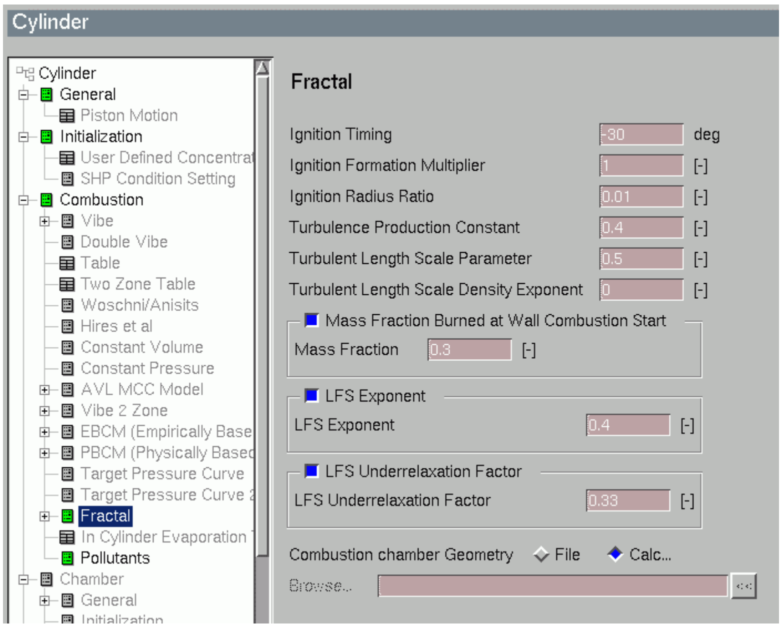


Figure 5—47: Fractal Combustion Model Window

Select the **Chamber Geometry calculation** folder to open the following window:

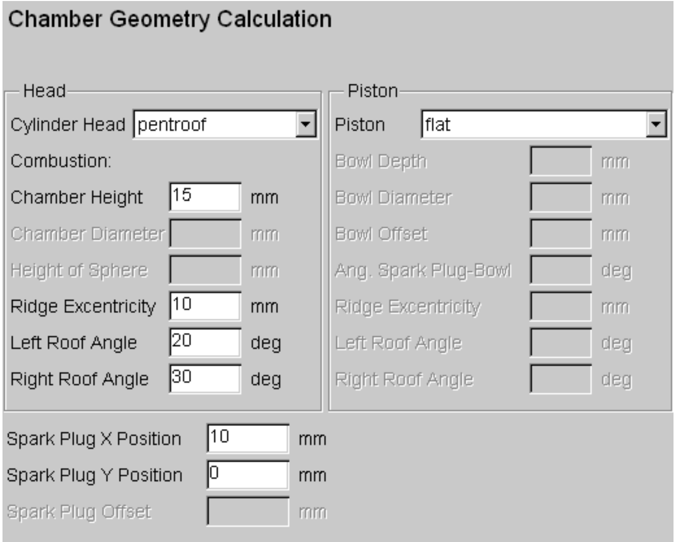


Figure 5—48: Chamber Geometry Calculation Window

5.5.2. 4t1fractal_species.bwf

Based on the `4t1fractal.bwf` example, the Fractal Combustion Model is used in combination with general species transport. Detailed pollutant formation chemistry is applied during the combustion in the burned zone.

5.5.3. 4t1calc_SHP_1zone.bwf

Based on the `4t1calc.bwf` example (refer to Figure 5—95), the ROHR is calculated using the Target Pressure Curve (1 zone) combustion model combined with the SHP conditions setting option.

In the **Cylinder | Properties | Initialization** window activate the **SHP Condition Setting** option and then in the tree select **SHP Condition Setting** to access the input fields.

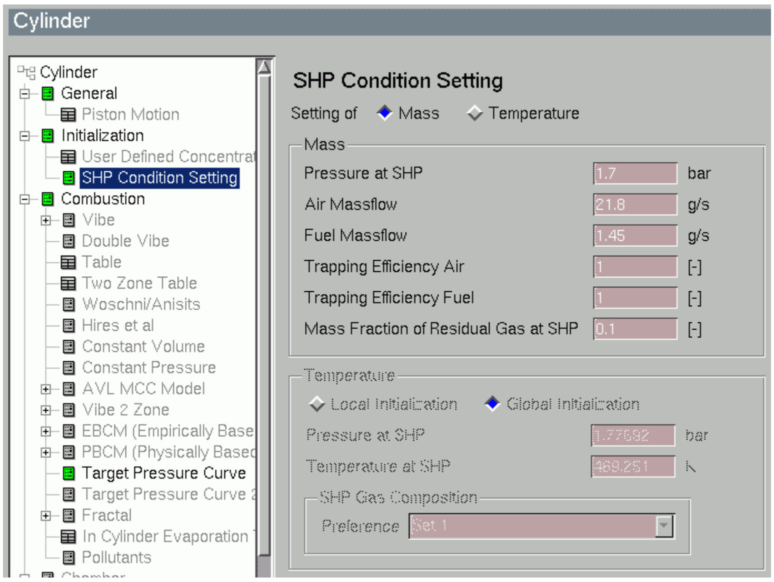


Figure 5—49: SHP Condition Setting Window

In the **Cylinder | Combustion** window select **Target Pressure Curve** from the **Heat Release** pull-down menu. Then select **Target Pressure Curve** in the tree to access the input fields:

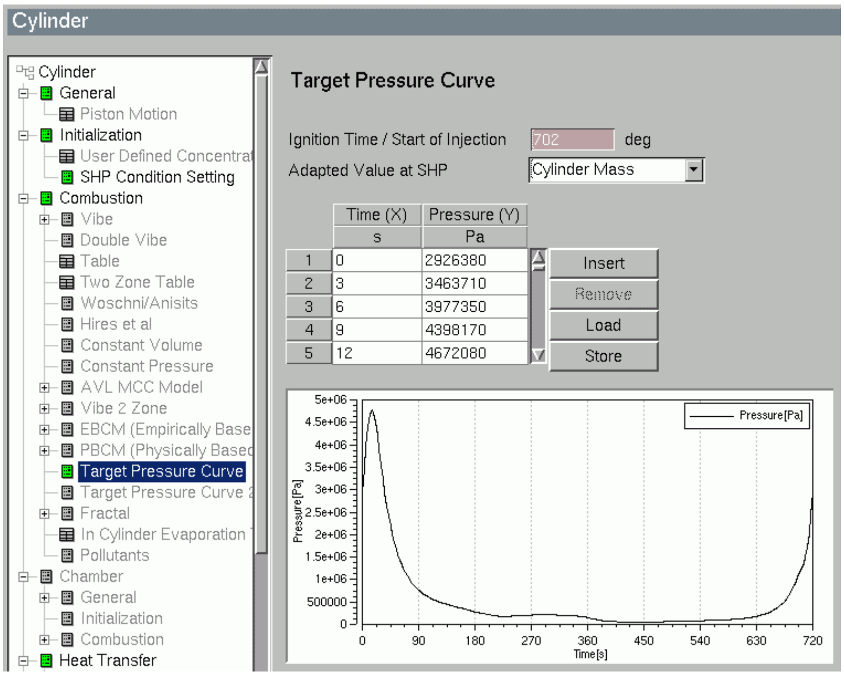


Figure 5—50: Target Pressure Curve Window

5.5.4. 4t1calc_SHP_2zone.bwf

Based on the `4t1calc.bwf` example (refer to Figure 5—95), the ROHR is calculated using the Target Pressure Curve 2 Zone combustion model combined with the SHP conditions setting option.

In the **Cylinder | Properties | Initialization** window activate the **SHP Condition Setting** option. Select **SHP Condition Setting** to access the fields shown in Figure 5—49. Select **Temperature**, then **Local Initialization** and enter the required data.

In the **Cylinder | Combustion** window select **Target Pressure Curve 2 Zone** from the **Heat Release** pull-down menu to access the fields shown in Figure 5—50.

5.5.5. 4t1calc_species_SHP_1zone.bwf

Based on the `4t1calc_species.bwf` example (refer to Figure 5—95), the ROHR is calculated using the Target Pressure Curve (1 zone) combustion model combined with the SHP conditions setting option.

In the **Cylinder | Properties | Initialization** window activate the **SHP Condition Setting** option. Select **SHP Condition Setting** to access the fields shown in Figure 5—49. Select **Temperature**, then **Global Initialization** and enter the required data.

The Fraction Set 1 chosen for the SHP Gas Composition refers to the **Globals | Initialization Mass Fraction** window:

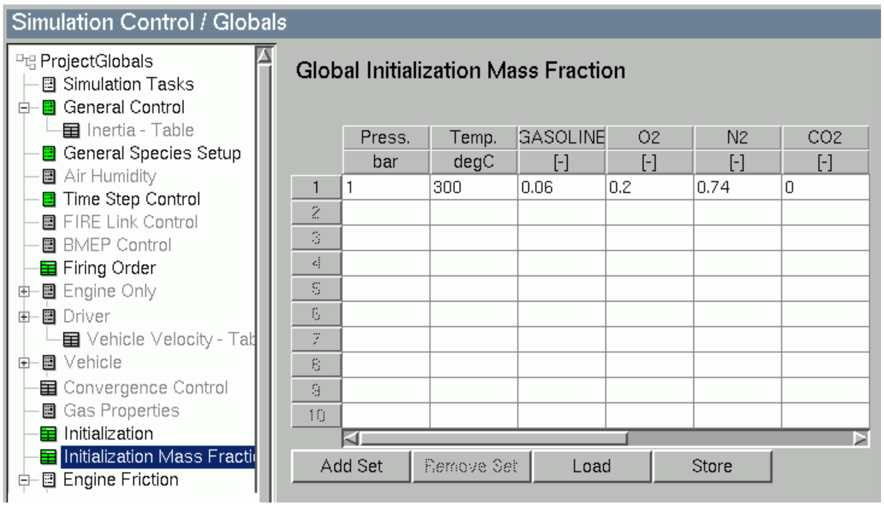


Figure 5—51: Globals / Initialization Mass Fraction

In the **Cylinder | Combustion** window select **Target Pressure Curve** from the **Heat Release** pull-down menu to open a window similar to Figure 5—50.

5.5.6. 4t1calc_species_SHP_2zone.bwf

Based on the `4t1calc_species.bwf` example (refer to Figure 5—95), the ROHR is calculated using the Target Pressure Curve 2 Zone combustion model combined with the SHP conditions setting option.

In the **Cylinder | Properties | Initialization** window activate the **SHP Condition Setting** option. Select **SHP Condition Setting** to access the fields shown in Figure 5—49 and Figure 5—51.

In the **Cylinder | Combustion** window select **Target Pressure Curve 2 Zone** from the **Heat Release** pull-down menu to open a window similar to Figure 5—50.

5.5.7. tcimcc.bwf

The combustion of the TCI engine is calculated with the AVL MCC model with default parameters valid for many types of engines. The combustion of the original `tcicalc.bst` is sharper which can be modeled by an increase of the combustion constant.

Based on the `tcicalc.bwf` example (refer to Figure 4—1), the AVL MCC Model is used.

In the **Cylinder | Properties | Combustion** window select **AVL MCC Model** from the Heat Release pull-down menu. Then select the **AVL MCC Model** sub-folder to open the following window:

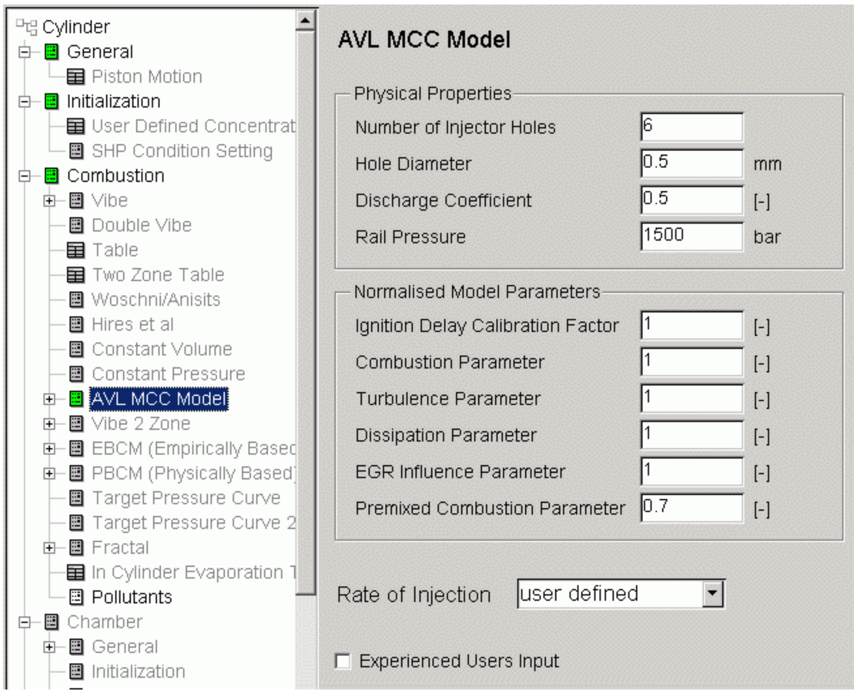


Figure 5—52: AVL MCC Model Window

Select the **Normalized ROI** folder to open the following window:

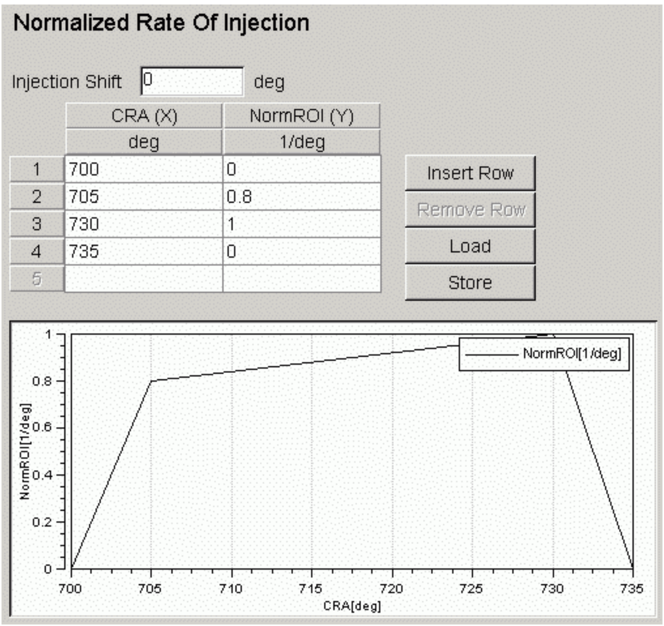


Figure 5—53: Normalized ROI Window

5.5.8. tcimcc_species.bwf

Based on the `tcimcc.bwf` example the AVL MCC Model is used in combination with general species transport.

5.5.9. tci_HCCI_skel_chem.bwf

The combustion of a TCI engine (based on the `tcimcc.bwf` example) is calculated with the “Single Zone HCCI” model. The skeletal mechanism for C7H16 by Barroso (see below) is used to simulate the auto-ignition and combustion process. It uses 26 chemical species and 66 chemical reactions.

The `tci_HCCI_skel_chem.bwf` was set-up such, that different levels of EGR are specified directly on the inlet boundary. Even if this is unrealistic for real-life engines, the model is still very useful to demonstrate the effect of different EGR levels on the processes of ignition and combustion. Figure 5—54 shows the heat release rates for 4 different cases:

- The ROHR profiles nicely show that first the cold flame ignition is taking place, after that the main ignition occurs.
- With increasing EGR the ignition delay increases. Cold flame and main ignition can not be distinguished any more.
- For very high levels of EGR (40% and 60%) the combustion is not complete, even if all fuel is consumed. A significant portion of the available heat is not released in the cylinder but is transported into the exhaust. A considerable increase in the mass fraction of CO can be seen

Reference:

- G. Barroso, “Chemical Kinetic Mechanism Reduction, Multizone and 3D-CRFD Modelling of Homogeneous Charge Compression Ignition Engines”, Dissertation, ETH, Zurich, 2006

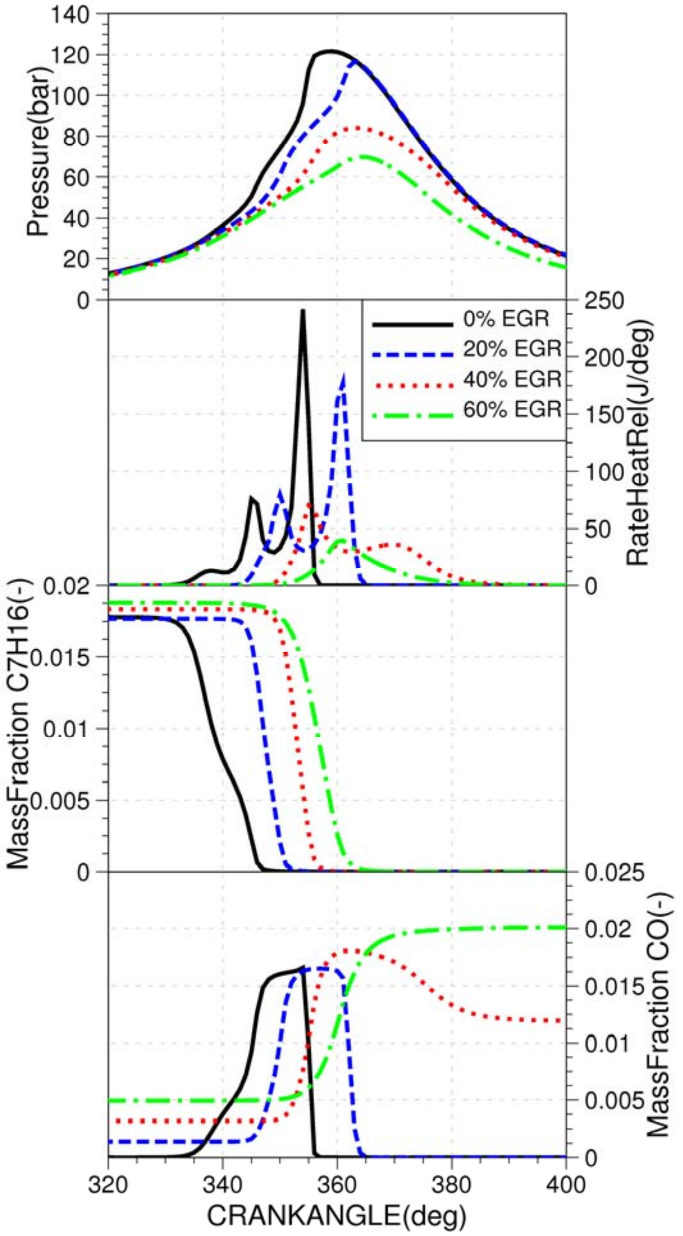


Figure 5—54: ROHR for different levels of EGR.

5.6. Compression Ignited

5.6.1. tcicalc.bwf

Refer to Chapter 4 for the example description.

This model is a 6-cylinder DI TCI truck engine. The model is somewhat simplified as the intake air cleaning and the exhaust silencing systems are not considered. This can be compensated by defining appropriate average pressures at the system boundaries 1 and 2 according to the expected pressure losses in the air cleaner and the back pressure due to the exhaust mufflers.

5.6.2. tcicalc_emotor.bwf

This model is derived from the model `tcicalc.bwf`. The Turbocharger is assisted by an Electrical Device to reach a Compressor Pressure Ratio of 2.5 (ECU controlled).

5.7. External

5.7.1. AVL CRUISE - ottocruise.bwf

By adding an ECU and modifying some Simulation Control/Globals settings of `ottocalc.bwf`, the example is prepared for a CRUISE–BOOST co-simulation.

5.7.2. AVL FIRE

5.7.2.1. r6o312.bwf

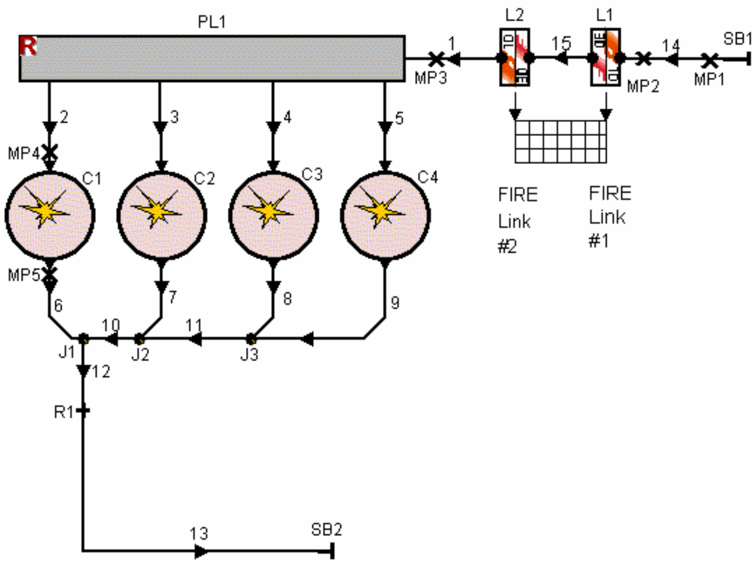


Figure 5—55: FIRE Link - The r6o312.bwf File

In this simplified model of a four cylinder four-stroke gasoline engine, a pipe section of the intake system is modeled applying the 3D CFD code AVL FIRE.

The three dimensional FIRE domain (`pipe.fpr`) is a straight axisymmetric pipe segment with a diameter of 56mm and a length of 200mm. The lengths of the two overlapping pipe sections (1D-3D) are 31.25 mm. The total number of 3D-cells in the FIRE domain is 21838.

5.7.2.2. FIRE_manifold.bwf

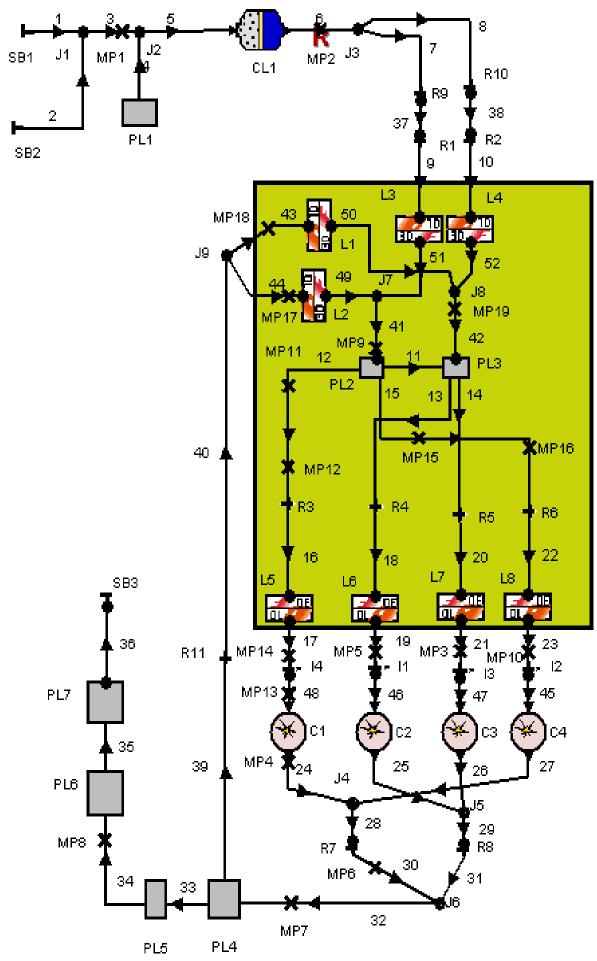


Figure 5—56: FIRE Link - The FIRE_manifold.bwf File

This is a model of a four cylinder four-stroke gasoline engine of a passenger car including a model for an EGR. In this model the whole intake manifold including the injectors is modeled applying the 3D CFD code AVL FIRE (`manifold.fpr`). The lengths of the eight overlapping pipe sections (1D-3D) are 10 mm. The total number of 3D cells in the FIRE domain is 861609.

5.7.3. CFD - 4t1calc_cfd_linked.bwf

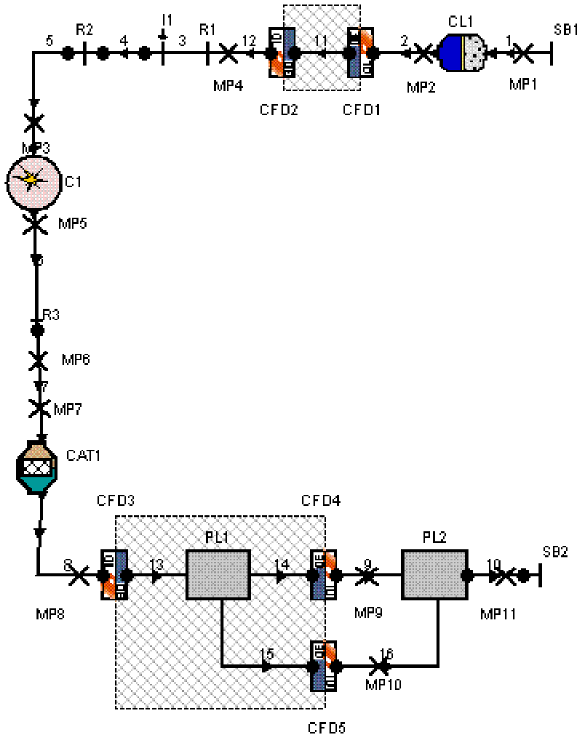


Figure 5—57: CFD Link – 4t1calc_cfd_linked.bwf

Based on the model 4t1calc.bwf parts of the intake and exhaust system are replaced by 3D domains which flow is simulated by a 3rd party CFD code.

5.7.4. User

These examples demonstrate the option to include models of elements and combustion using compiled source code. It is necessary to compile and link a new version of the **BOOST** executable to run these examples and use this feature.

5.7.4.1. 2plenums.bwf

This is a simple gas dynamic model to demonstrate the use of User Defined Elements (UDEs) in **BOOST**. The example consists of two plenums which are replicated in the user written code and are compared to the identical model that uses the plenum junction models available in **BOOST**.

Source files: *bst_userdef_element.f90, mod_userdef_element.f90, mod_userdefined.f90, userfefined.f90*

For more detailed information about compiling and linking a user defined BOOST executable please refer to the **BOOST** Interfaces Manual.mailto:boost_support@avl.com

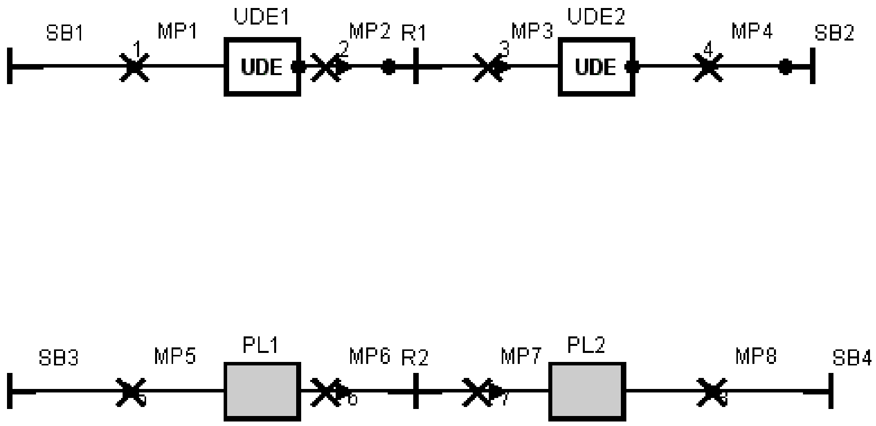


Figure 5—58: 2plenums.bwf File

5.7.4.2. 4t1userdef_hpc.bwf

Based on the `4t1calc.bwf` example (refer to Figure 5—95), a user defined high pressure cycle is selected.

In the **Cylinder | Properties | Combustion** window select **UD-High Pressure Cycle** from the Heat Release pull-down menu.

This example can be run using the BOOST executable from the installation. As example, the Vibe model is replicated. In order to make changes to the applied heat release model it is necessary to recompile BOOST using the static library and user written code. Therefore source files from the installation are required.

Source files: *cy_userdef_hpc.f90, mod_userdef_hpc.f90, mod_userdefined.f90, userfefined.f90*

For more detailed information about compiling and linking a user defined BOOST executable please refer to the **BOOST** Interfaces Manual.mailto:boost_support@avl.com

5.7.4.3. 4t1userdef_comb.bwf

Based on the `4t1calc.bwf` example (refer to Figure 5—95), a user defined Combustion (Rate of Heat Release) is selected.

In the **Cylinder | Properties | Combustion** window select **User Model** from the **Heat Release** pull-down menu.

This example can be run using the BOOST executable from the installation. As example, the Vibe model is replicated. In order to make changes to the applied heat release model it is necessary to recompile BOOST using the static library and user written code. Therefore source files from the installation are required.

Source files: *cy_userdef_comb.f90, mod_userdef_comb.f90, mod_userdefined.f90, userfefined.f90*

For more detailed information about compiling and linking a user defined BOOST executable please refer to the **BOOST** Interfaces Manual.mailto:boost_support@avl.com

5.8. Gas Dynamics

5.8.1. replcalc.bwf

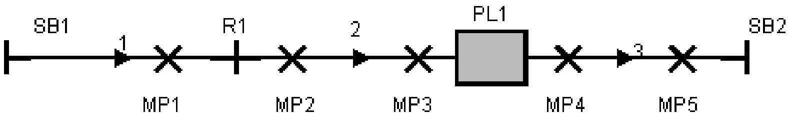


Figure 5—59: replcalc.bwf File

This file contains a simple model with some basic elements. Only one "cycle" is calculated. The boundary conditions are not in equilibrium with the initial conditions in the system. Pressure waves start to run through the system. After the calculation of several cycles steady state conditions will be reached.

5.8.2. laval.bwf

Model of a converging-diverging nozzle. The pressure at System Boundary 2 enforces a stationary shock front in the divergent part of the nozzle. The development of the shock front can be displayed by using spatial plots in IMPRESS Chart (Calculation Mode: Animation).

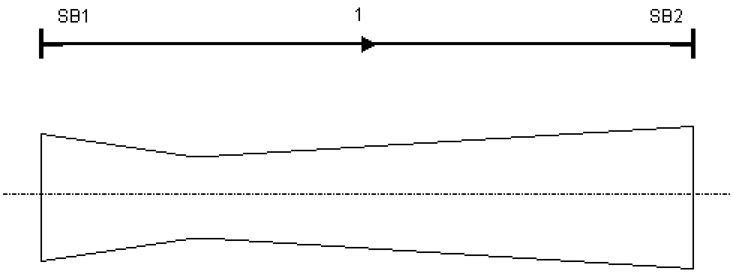


Figure 5—60: laval.bwf File

5.8.3. shock_tube.bwf

The shock tube test generates the simulation of a shock wave. This example is used and documented extensively in the literature on gas dynamics as a numerical experiment.

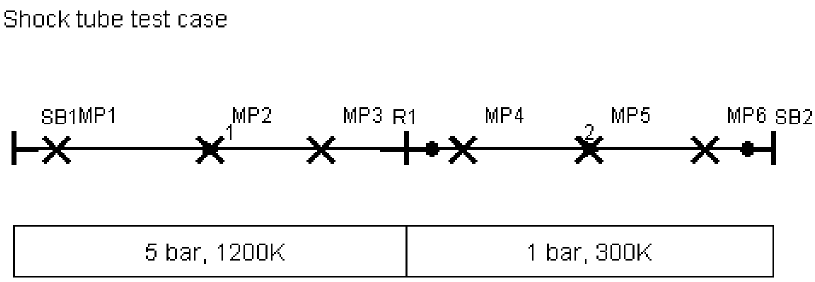


Figure 5—61: shock_tube.bwf File

5.8.4. shock_tube_reactive.bwf

This example simulates the transient behavior of a 12 cm long shock tube with chemical reactions (auto-ignition chemistry for H2/O2/Ar):

Initialization:

- Temperature: 624K, 600K and 500K (3 different Cases)
- Pressure: 0.366 bar
- Velocity: -478.5 m/s
- Mass Fraction H2: 0.012773
- Mass Fraction O2: 0.101369
- Mass Fraction Ar: 0.885858

Boundary Conditions:

- left boundary: closed
- right boundary: open

Reference:

- B. Erdem, "Finite Volume Solutions of 1D Euler Equations for High Speed Flows with Finite Rate Chemistry", MSc Thesis, Middle East Technical University, Turkey, 2003

Resuts:

- After the start of the simulation, a shock is formed at the (closed) left boundary; behind the shock the pressure is 1.31 bar, the temperature is 1036K.
- The shock is moving to the right.
- Behind the shock the temperature is high enough to initiate auto-ignition of the H2/O2 mixture,
- After an ignition delay a second shock is formed (visible at 150μs).
- The second shock, resulting from the combustion of H2, is moving to the right at a higher speed than the first shock and passes at approximately 180 μs.
- With decreasing initial temperatures (Case_2 and Case_3) the ignition delay increases. For Case_3 ignition does not occur.

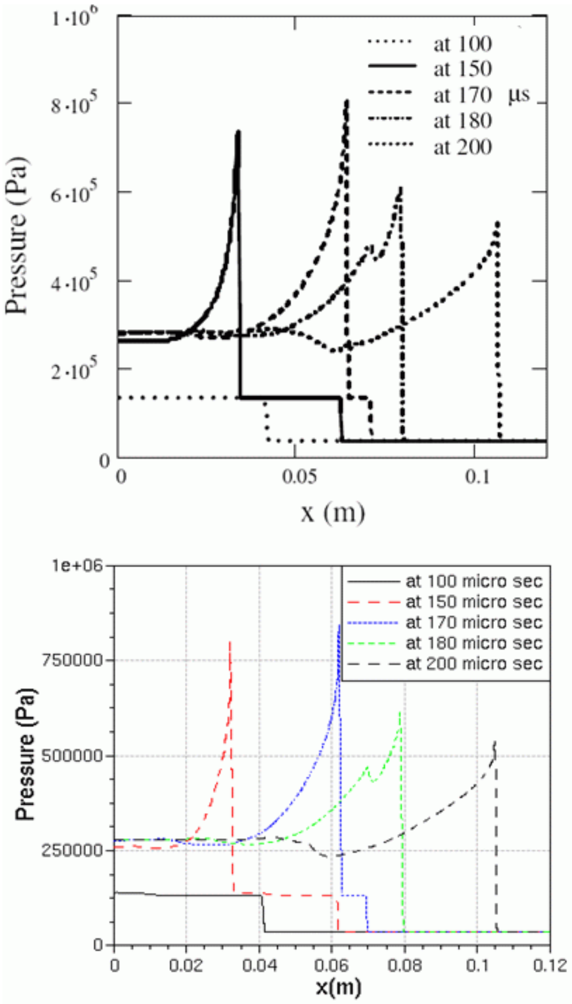


Figure 5—62: Results of shock_tube_reactive.bwf Case_1. Reference and BOOST (below).

5.9. MATLAB

5.9.1. Application Programming Interface (API)

5.9.1.1. c_4t1calc_api_mdl.bwf, c_4t1calc_api_m.bwf

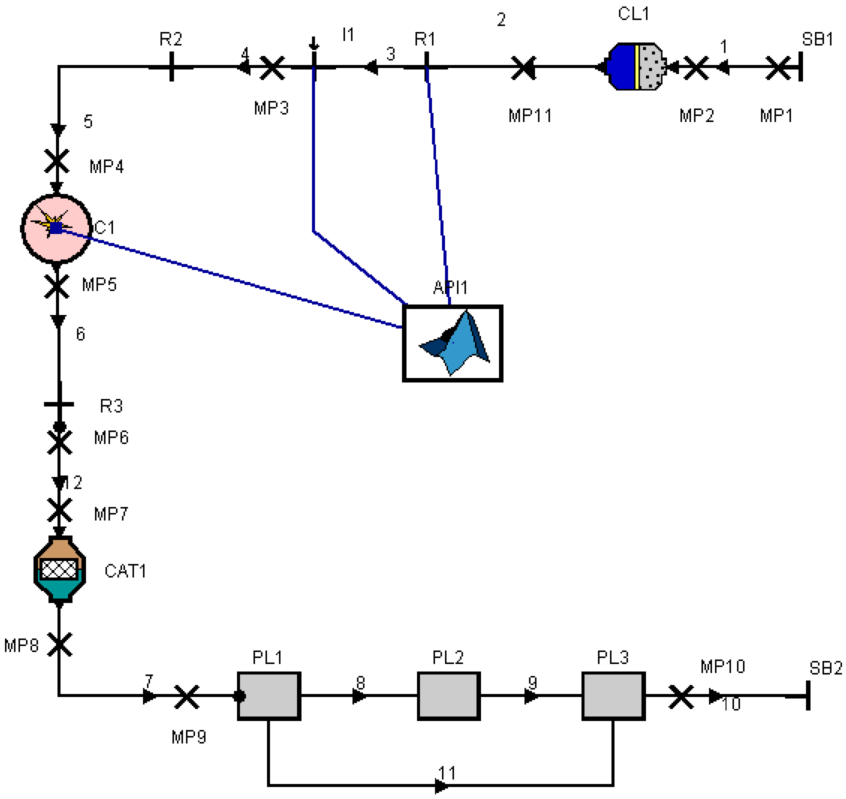


Figure 5—63: Single Cylinder Model with Links to MATLAB API

A speed course control using the MATLAB-API element was added to the example `4t1calc.bwf`.

For c_`4t1calc_mdl.bwf` this control is performed by the MATLAB-Simulink model `c_4t1calc_api.mdl` while `c_4t1calc_m.bwf` uses the MATLAB-m-function `c_4t1calc_api_m.m (where c_4t1calc_api_m_setup.m` is necessary to initialize the m-function example). For plotting the channel value evolution of `4t1calc_mdl.bwf` two additional m-function files (`c_4t1calc_api_startup.m`, `c_4t1calc_api_stop_fcn.m`) were introduced.

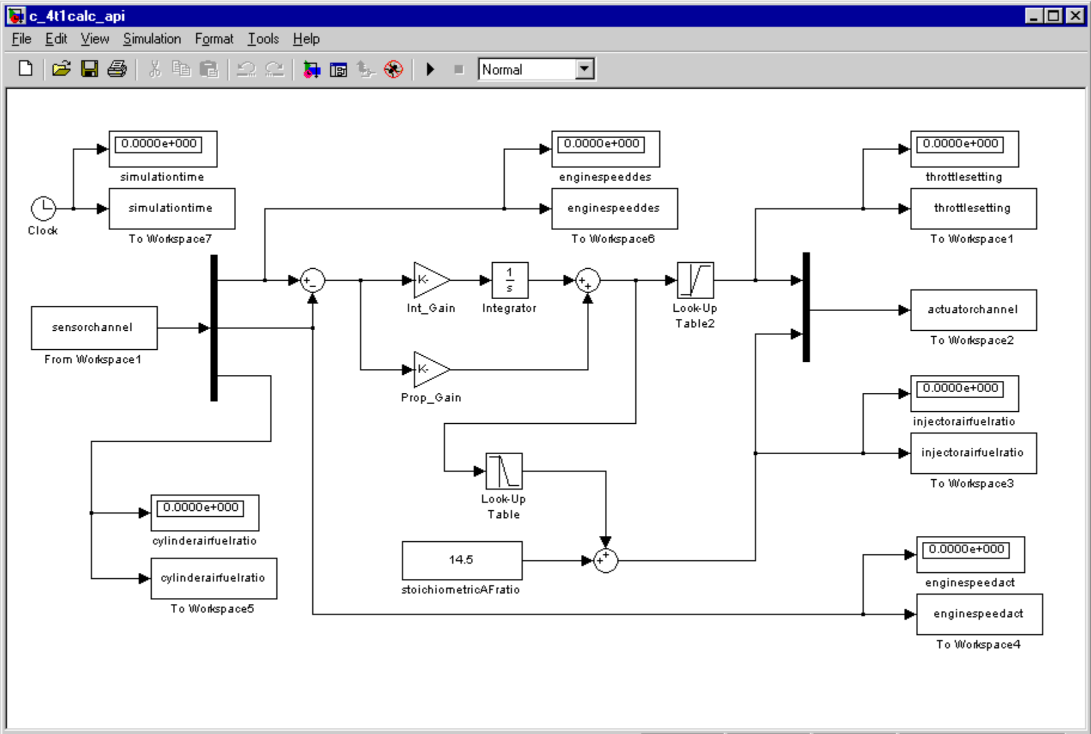


Figure 5—64: MATLAB-Simulink Model c_4t1calc_api.mdl

5.9.2. MATLAB Dynamic Link Library (DLL)

5.9.2.1. 4t1calc_dll.bwf

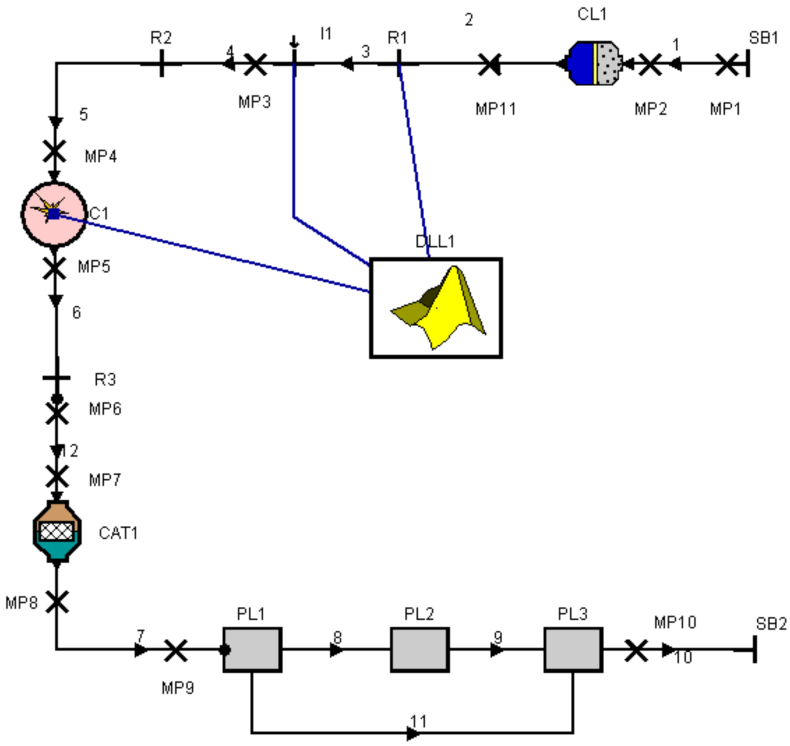


Figure 5—65: Single Cylinder Model with Links to MATLAB DLL

A speed course control using the MATLAB-DLL element was added to the example `4t1calc.bwf`.

This control is performed by a MATLAB-Simulink Real Time Workshop generated dynamic link library based on the MATLAB-Simulink model `c_4t1calc_dll.mdl`.

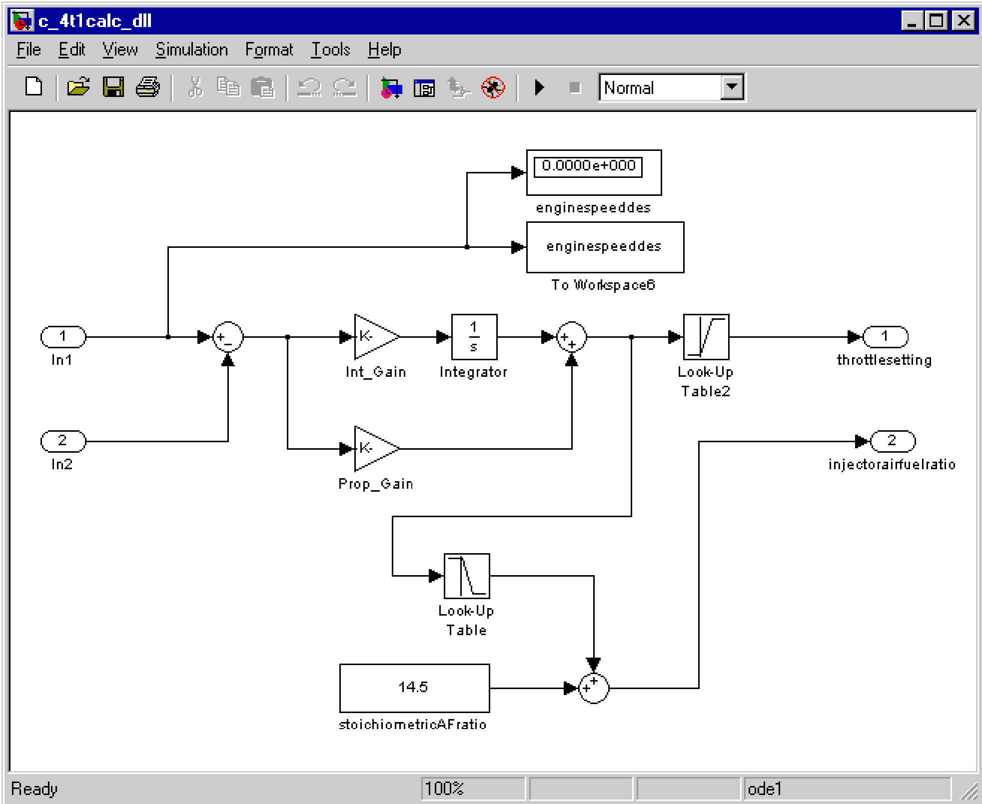


Figure 5—66: MATLAB -Simulink Model c_4t1calc_dll.mdl

5.9.3. Code Generation

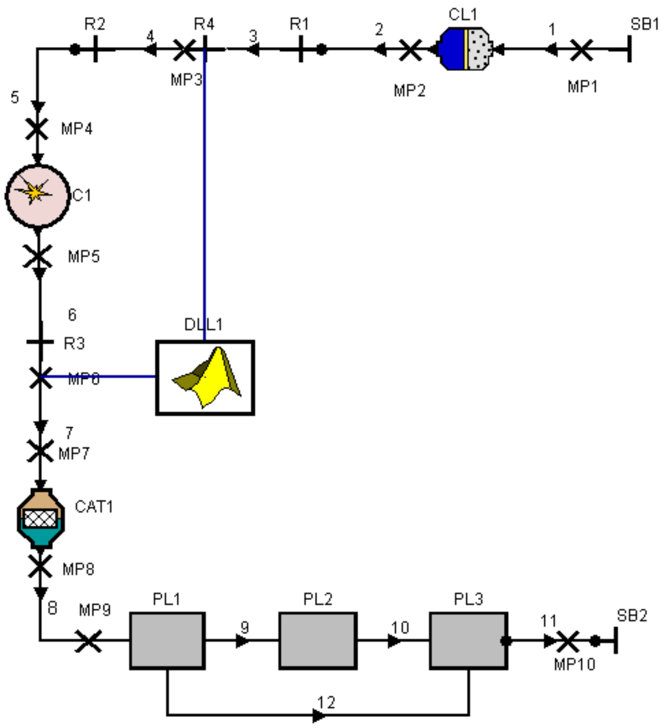


Figure 5—67: BOOST Model for Both Code Generation Examples

This example is repeated once in C (4t1_c_dll) and then in FORTRAN 90 (4t1_f90_dll). Two sensors are set to measure first the BOOST time and secondly the pressure at Measuring Point 6. One actuator is used to set the flow coefficient of Restriction 4. The stepSize is set to 0.01 so BOOST and the DLL interact every 0.01 seconds after the initial settling period of 3 cycles. Each called routine prints a message to the screen to show that it has been called. For the step function (`_mdlInterfaceStep`) this is only done once (the first time it is called). When the DLL step function is called it simply sets the flow coefficient for the restriction to 0.9 and displays the values of the sensors.

Note that there is little error checking in the DLL and it is setup purely for this example where the numbers and types of the channels all match.

5.9.3.1. 4t1_c_dll.bwf

This example calls the DLL generated from code in C (*dll_for_boost.c*). The code has been written explicitly for this example, Windows and the Visual C++ compiler (6.0). A Visual Studio project is provided for this example (*c_code_dll.dsp*) and a pre compiled Windows DLL.

5.9.3.2. 4t1_f90_dll.bwf

This example calls the DLL generated from code in FORTRAN 90 (*dll_for_boost.f90*). The code has been written explicitly for this example, Windows and the Visual Compaq compiler (version 6.6B). A Visual Studio project is provided for this example (*f_code_dll.dsp*) and a pre compiled Windows DLL.

5.9.4. System Function (s-function) in BOOST Gas Exchange

5.9.4.1. monitor.bwf

Three wires and a MATLAB DLL element have been added to the example `4t1calc.bwf`. The s-function link is selected for the MATLAB DLL element and for each of the wires a sensor channel for the pressure at the attached measuring point is selected. This example, therefore, simply monitors the pressure at three points in the model and changes nothing (no actuators).

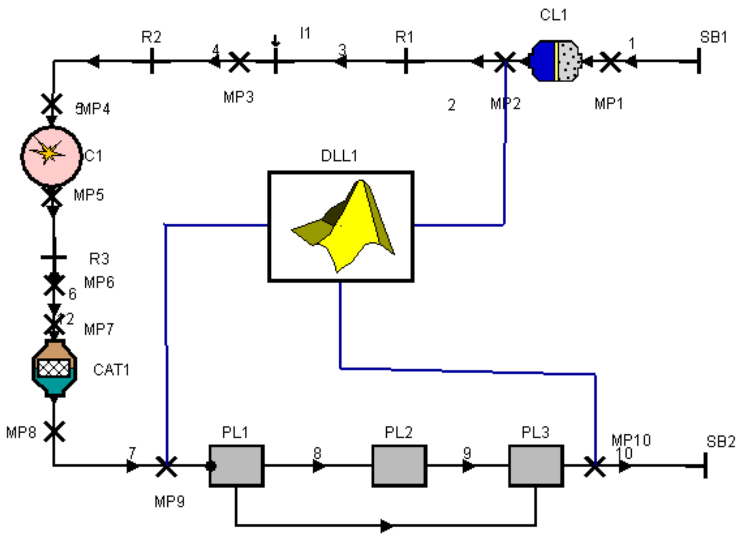


Figure 5—68: BOOST Model for the Monitor Example

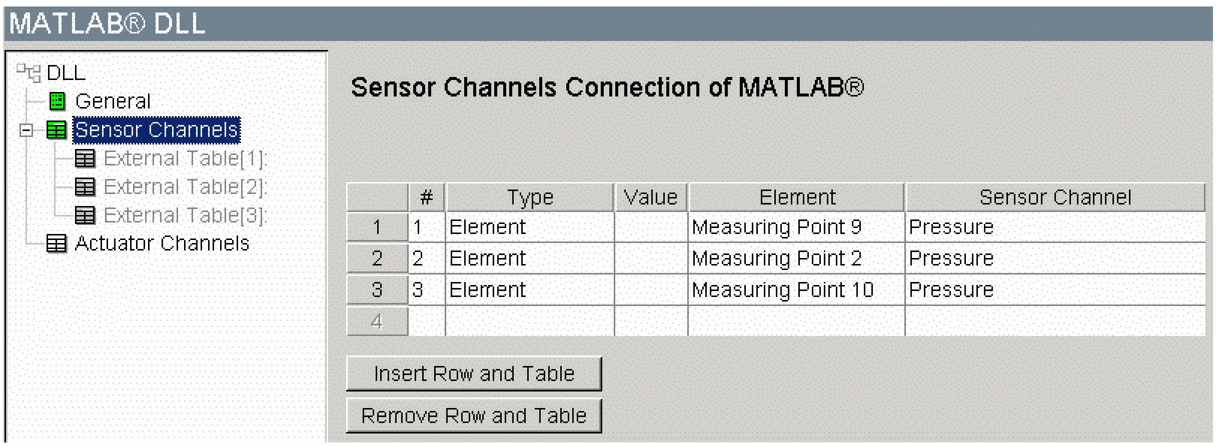


Figure 5—69: monitor.bwf - Sensor Channels

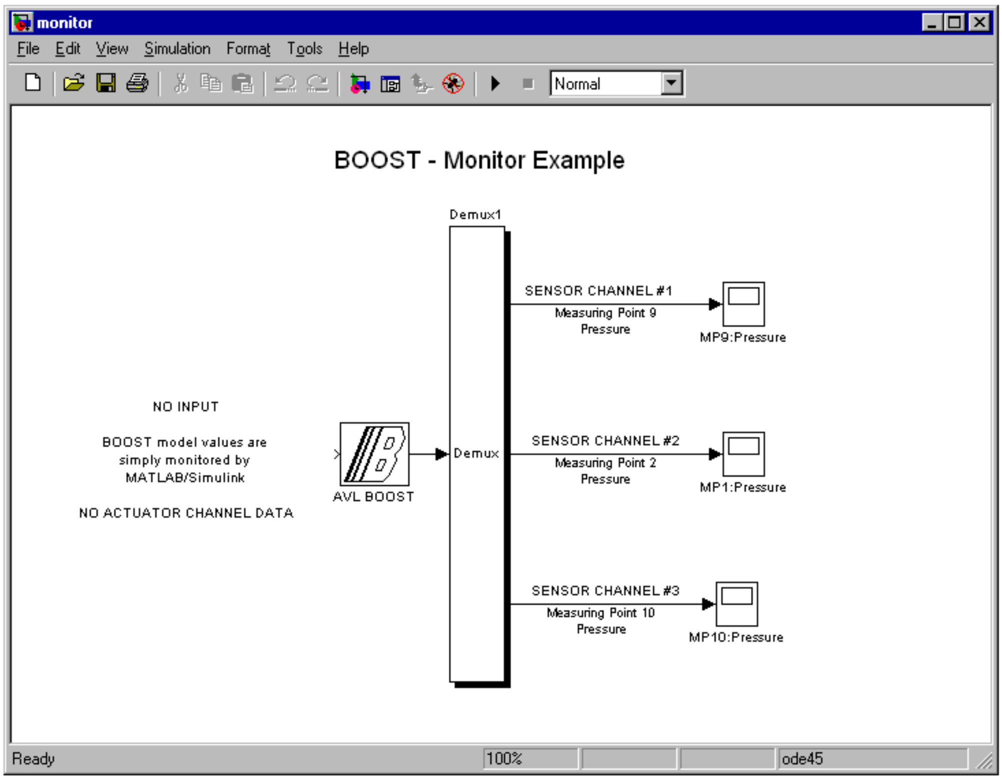


Figure 5—70: MATLAB-Simulink Model for the Monitor Example

5.9.4.2. af_control.bwf

The MATLAB/Simulink model is used to control the air/fuel ratio of an injector based on the logic in the model and the sensor value of the air/fuel ratio in the cylinder. In this simple example the air/fuel ratio at the injector is changed until the target value of 14.5 (air/fuel ratio in the cylinder) is reached.

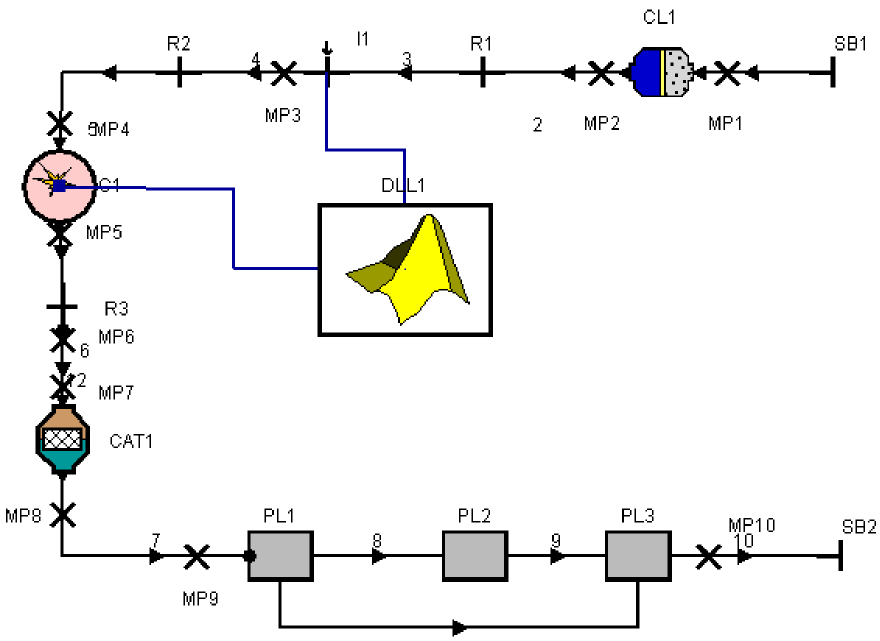


Figure 5—71: BOOST Model for the af_control Example

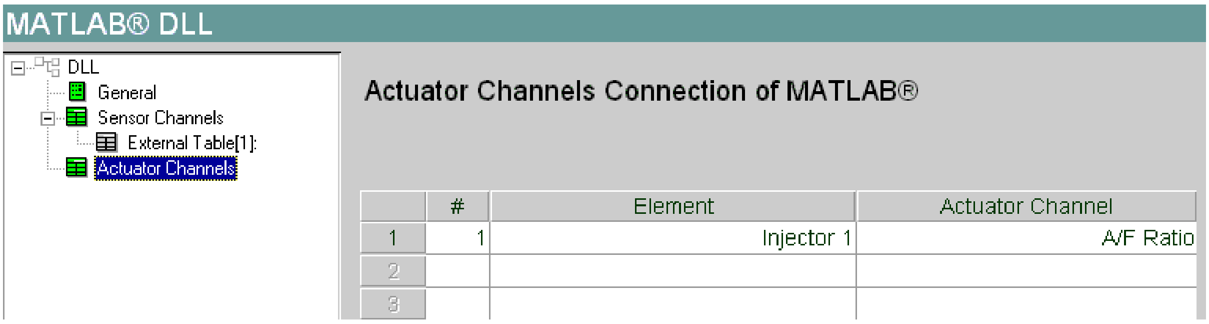


Figure 5—72: af_control.bwf - Actuator Channels

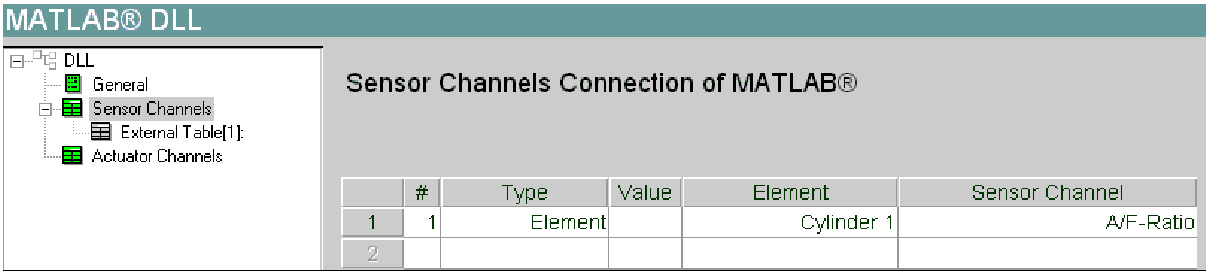


Figure 5—73: af_control.bwf - Sensor Channels

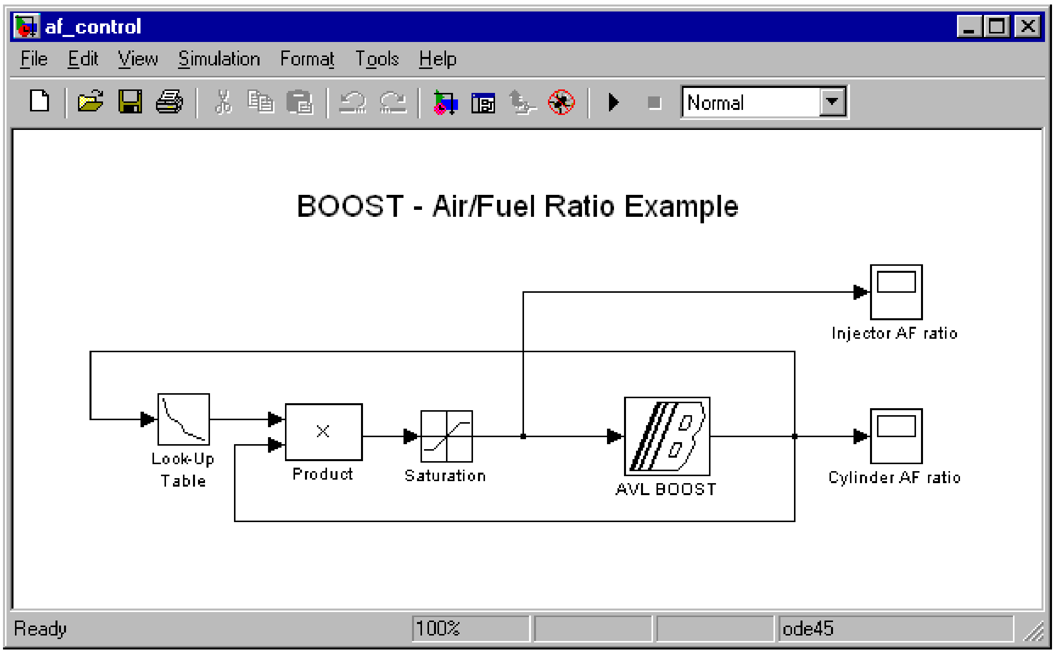


Figure 5—74: MATLAB-Simulink Model for the af_control Example

5.9.4.3. egr_control.bwf

The MATLAB/Simulink model is used to control the opening of a restriction (model of an EGR valve) based on the average value of residual gas in the intake runners. The target EGR value for the control logic is 10%.

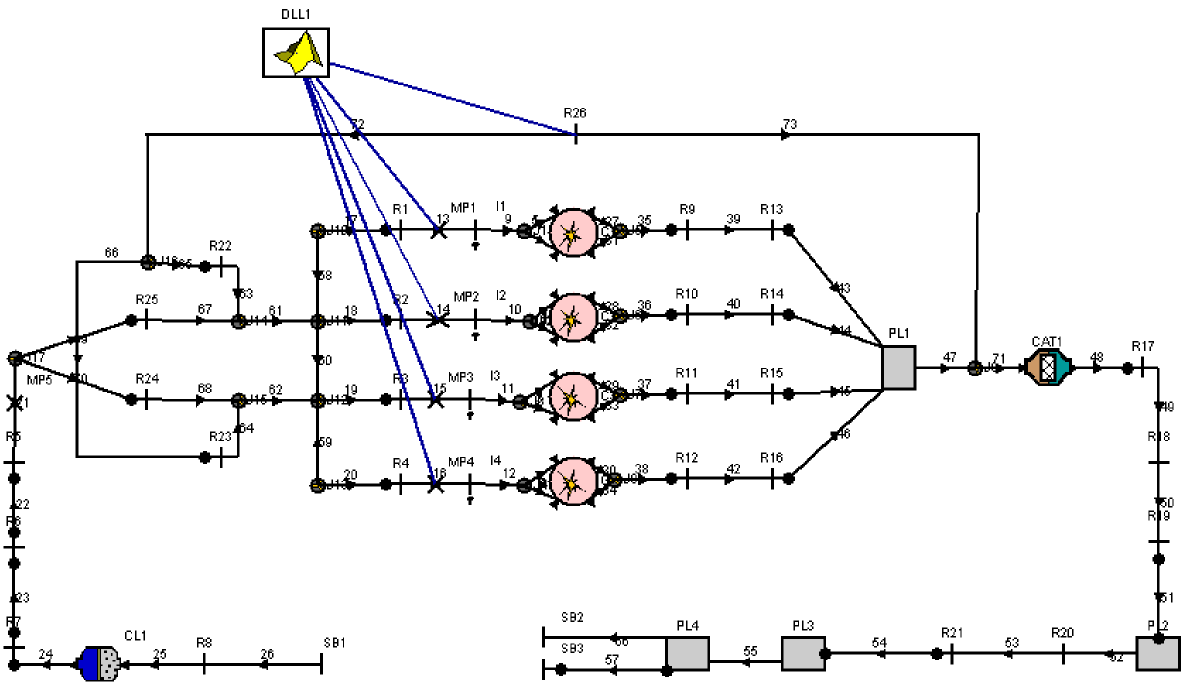


Figure 5—75: egr_control.bwf File

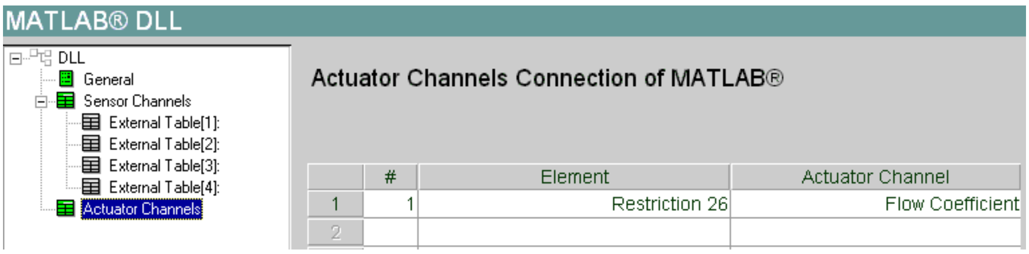


Figure 5—76: egr_control.bwf - Actuator Channels

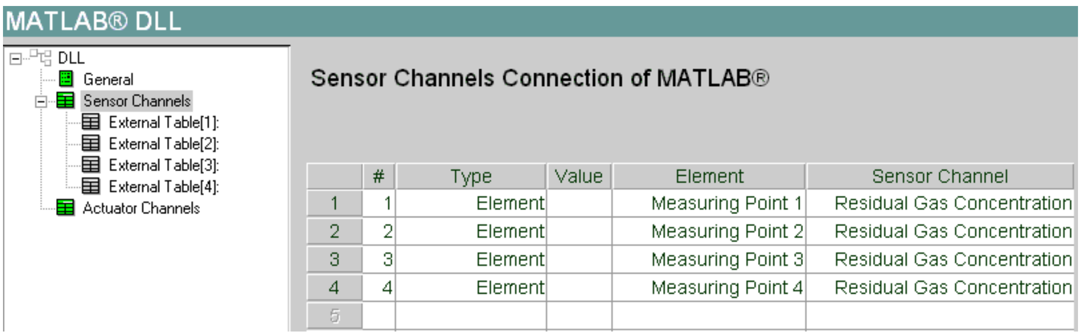


Figure 5—77: egr_control.bwf - Sensor Channels

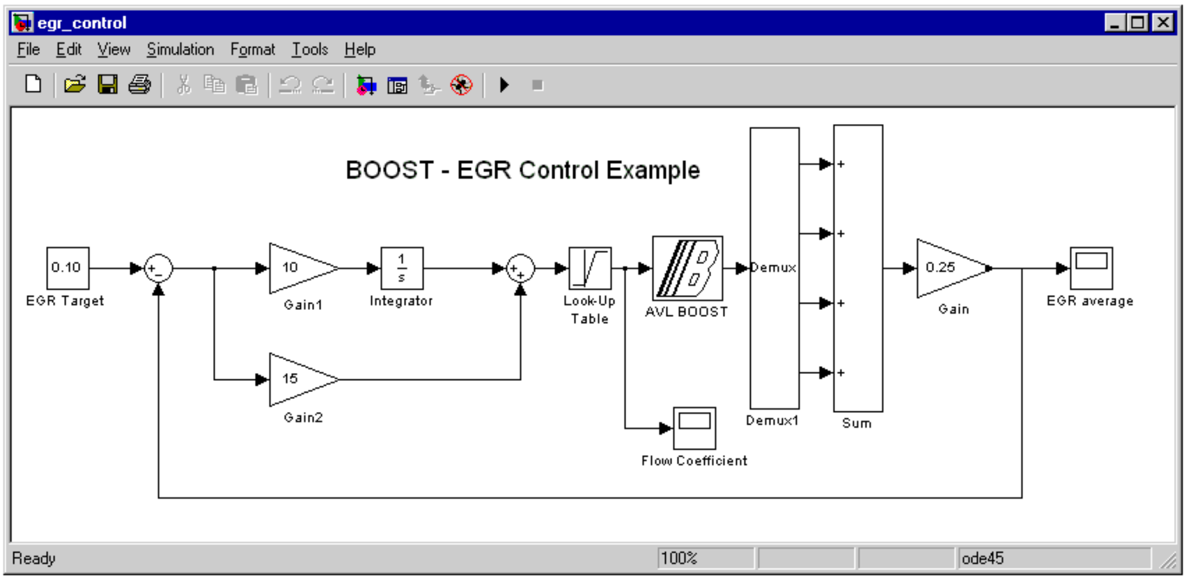


Figure 5—78: egr_control.mdl File

5.9.4.4. boundary_control.bwf

The MATLAB/Simulink model is used to control the internal boundaries of a single cylinder BOOST model. The internal boundaries represent the intake and exhaust system. The pressure, temperature and species content of the boundaries are set (actuated) from the MATLAB/Simulink model. The intake pressure is obtained from a look-up table in the MATLAB model. The pressure is defined against crank angle, which is obtained from the global crank angle sensor of the BOOST model. The duration of the simulation is also controlled by MATLAB based on the absolute crank angle sensor so that the BOOST model runs at least 50 cycles. After 0.5 seconds the EGR is stepped from 0% to 15% by actuating the residual gas concentration of internal boundary 1 (intake boundary).

Table 5-1: Boundary Control Actuators

#	Element	Actuator Channel
1	Internal Boundary 1	Pressure
2	Internal Boundary 1	Temperature
3	Internal Boundary 1	Residual gas concentration
4	Internal Boundary 1	Fuel concentration
5	Internal Boundary 1	A/F ratio
6	Internal Boundary 2	Temperature
7	Internal Boundary 2	Residual gas concentration
8	Internal Boundary 2	Fuel concentration
9	Internal Boundary 2	A/F ratio

Table 5-2: Boundary Control Sensors

#	Element	Sensor Channel
1	Cylinder 1	Pressure
2	Cylinder 1	Temperature
3	Global	Crank angle
4	Global	Absolute crank angle

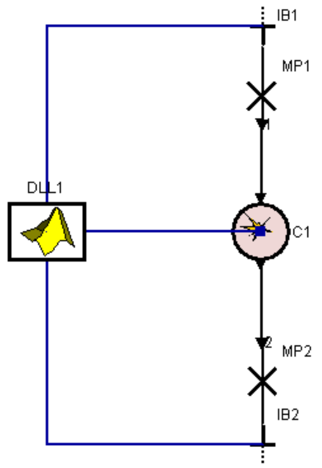


Figure 5—79: boundary_control.bwf File

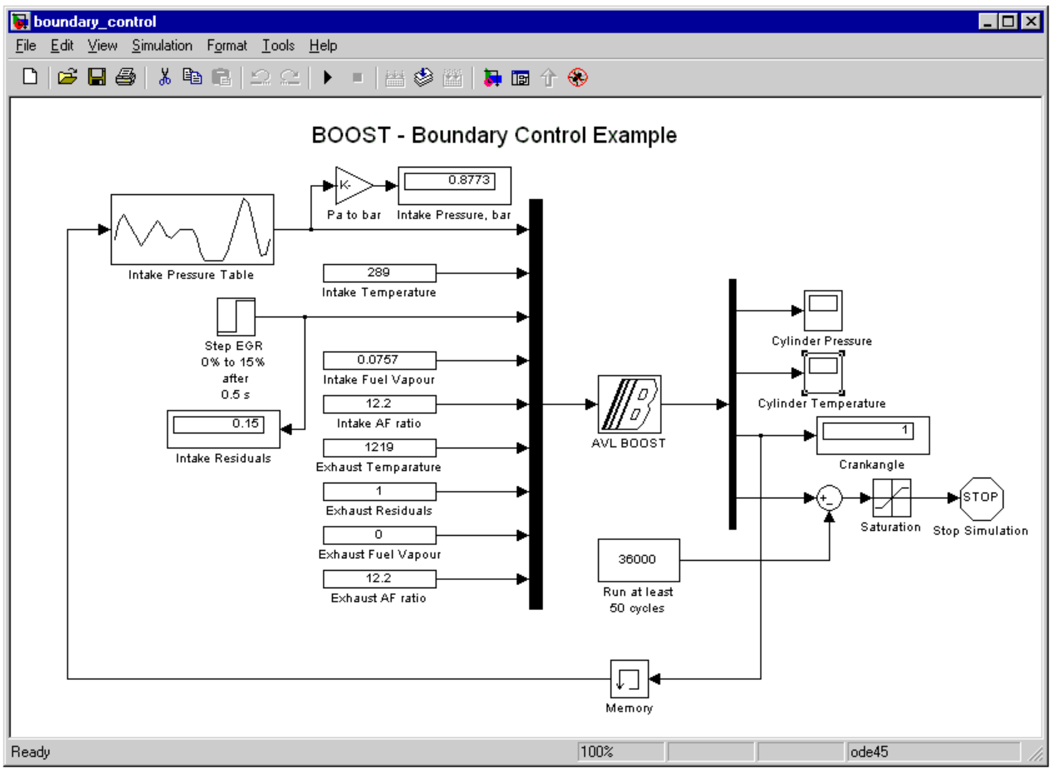


Figure 5—80: boundary_control.mdl File

5.9.4.5. cylinder_monitor.bwf

The MATLAB/Simulink model is used to monitor the pressure and temperature of each cylinder a four cylinder BOOST model. The maximum pressure and temperature of cylinder 1 is saved and displayed in the MATLAB/Simulink model.

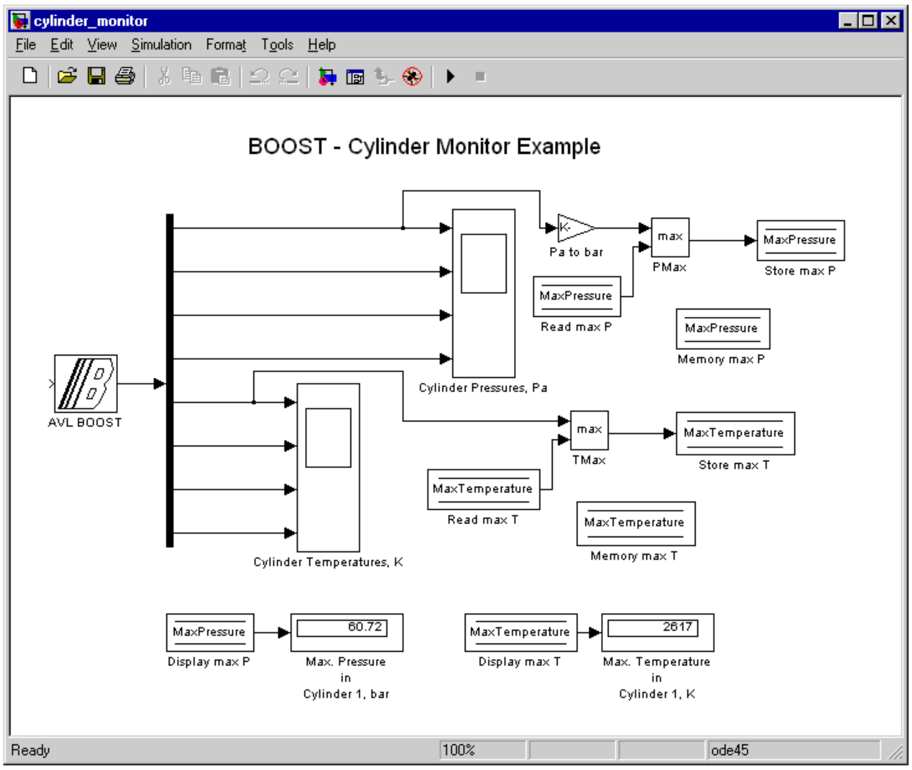


Figure 5—81: cylinder_monitor.mdl File

5.9.5. System Function (s-function) in BOOST Aftertreatment

5.9.5.1. DOC1_DPF1__DOC.bwf, DOC1_DPF1__DPF.bwf

This example shows how a DOC-DPF system is simulated with BOOST and Simulink. Therefore two individual BOOST models are setup and applied in Simulink. Details of the model are also given by the example of Section 5.2.18.

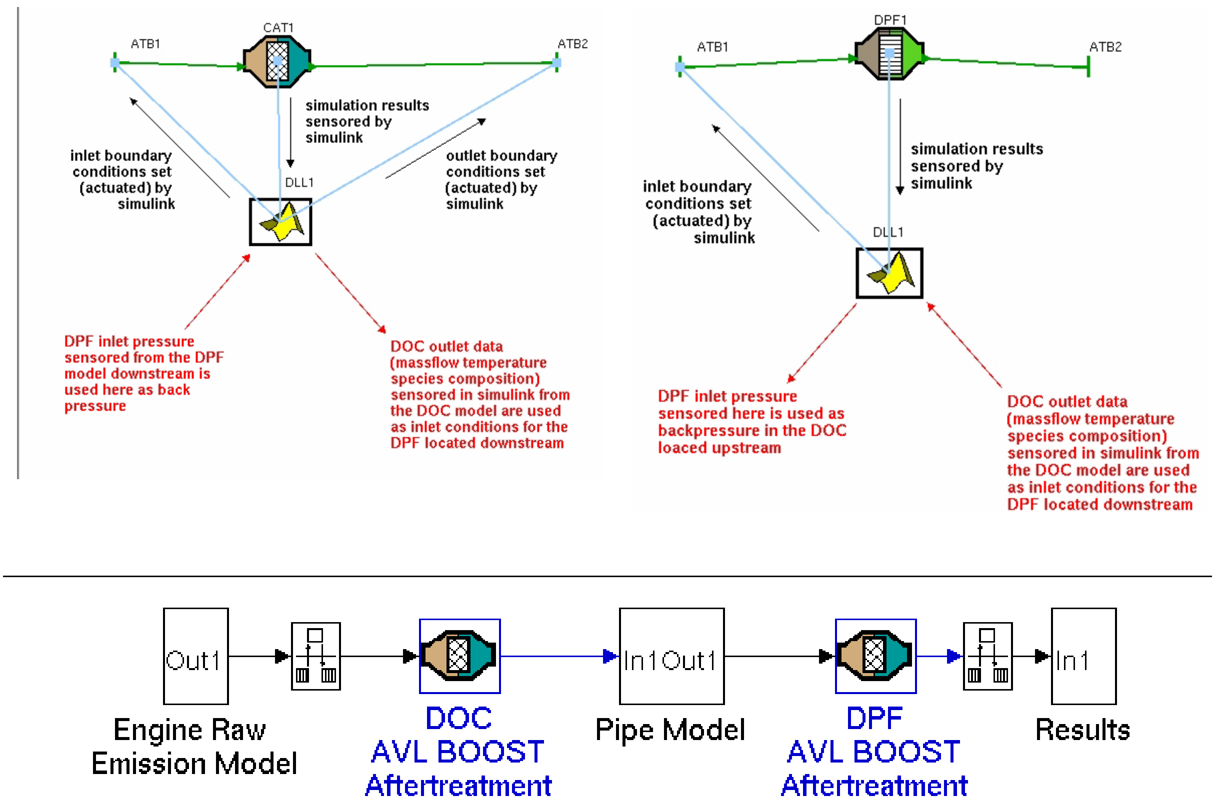


Figure 5—82: BOOST Input and Simulink Model of a DOC-DPF System

5.9.5.2. DOC1_DPF1__SCR.bwf

This example shows how a DOC-SCR system is simulated with BOOST and Simulink. Therefore one BOOST model is setup and applied in Simulink. Details of the model are also given by the example of Section 5.2.15.

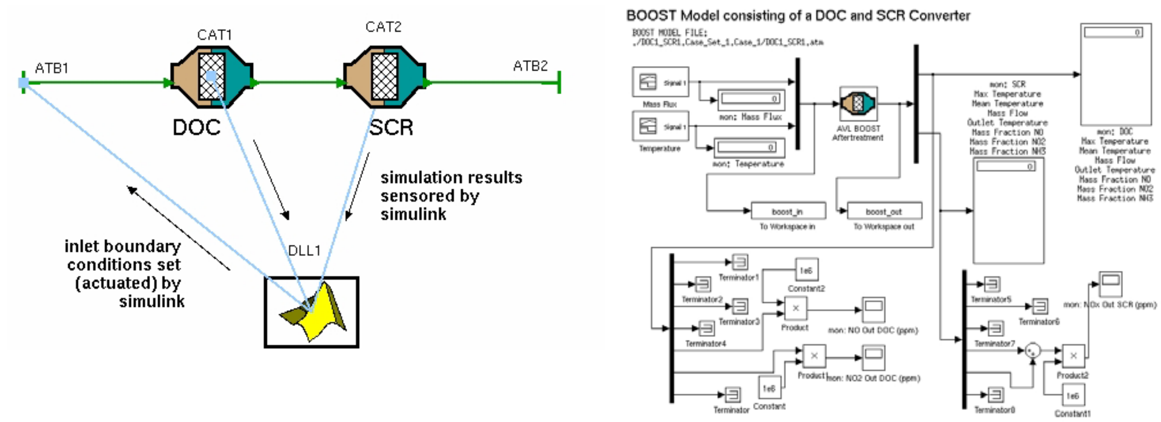


Figure 5—83: BOOST Input and Simulink Model of a DOC-SCR System

5.9.5.3. DPF_BareTrap_Regeneration.bwf

This example shows the simulation of a DPF bare trap regeneration in BOOST and Simulink. Therefore one BOOST model is setup and applied in Simulink. Details of the model are also given by the example of Section 5.2.12.

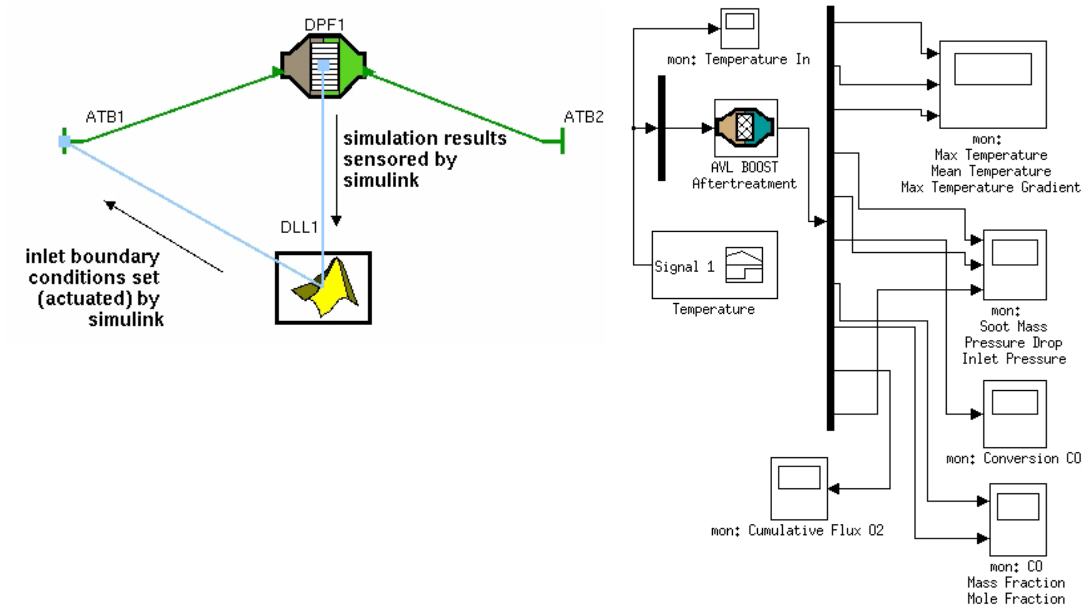


Figure 5—84: BOOST Input and Simulink Model of a DPF Bare Trap Regeneration Simulation

5.9.5.4. OxiCat_LightOff.bwf

This example shows how a DOC light-off is simulated with BOOST and Simulink. Therefore one BOOST model is setup and applied in Simulink. Details of the model are also given by the example of Section 5.2.4.

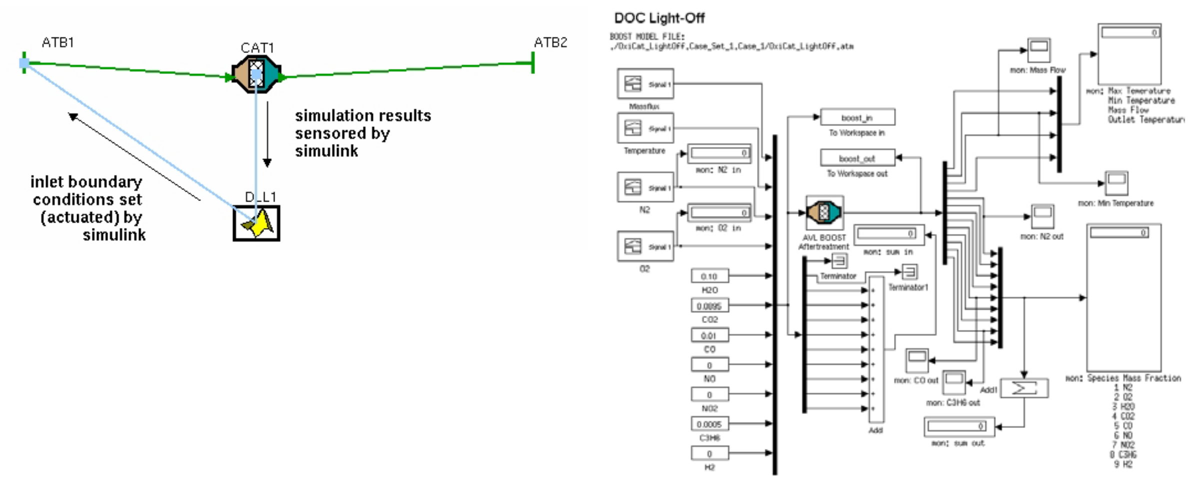


Figure 5—85: BOOST Input and Simulink Model of a DOC Light-Off Simulation

5.9.5.5. SCRCat_Trans_AdDesorption.bwf

This example shows how NH3 ad-desorption in a SCR catalyst is simulated with BOOST and Simulink. Therefore one BOOST model is setup and applied in Simulink. Details of the model are also given by the example of Section 5.2.10.

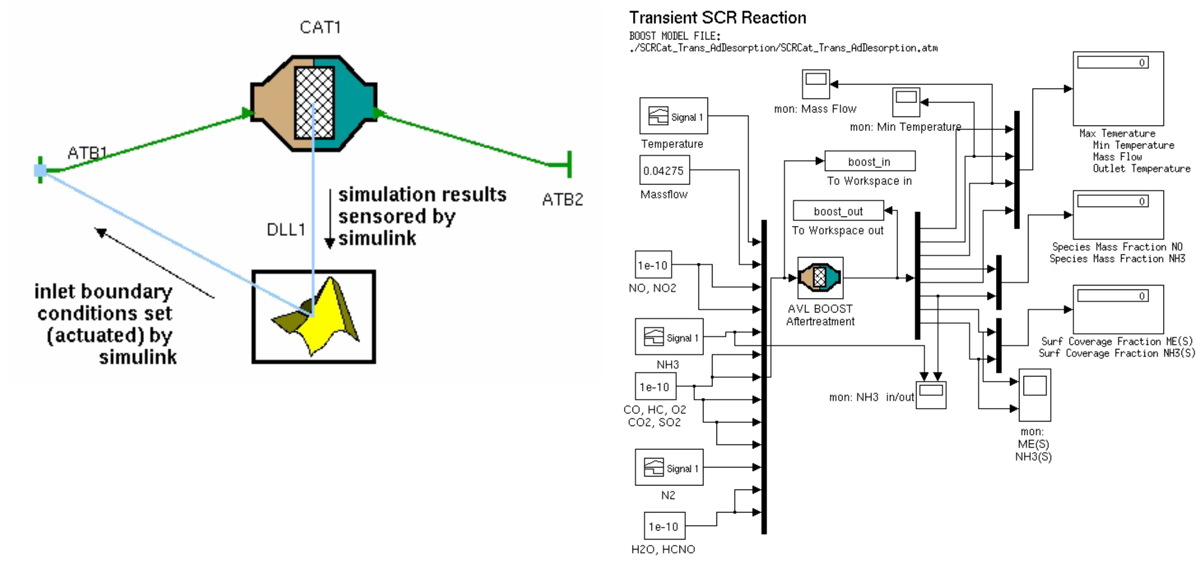


Figure 5—86: BOOST Input and Simulink Model of a SCR Simulation Using the Transient Kinetic Approach

5.10. Other

5.10.1. L3_1.2L_TCI_DI_Diesel_Engine.bwf

This is the model of a turbo charged direct injection Diesel engine, featuring a turbo charger for calculation with the simplified TC algorithm and a model for an EGR valve. The model was created for the determination of the full load performance of the engine.

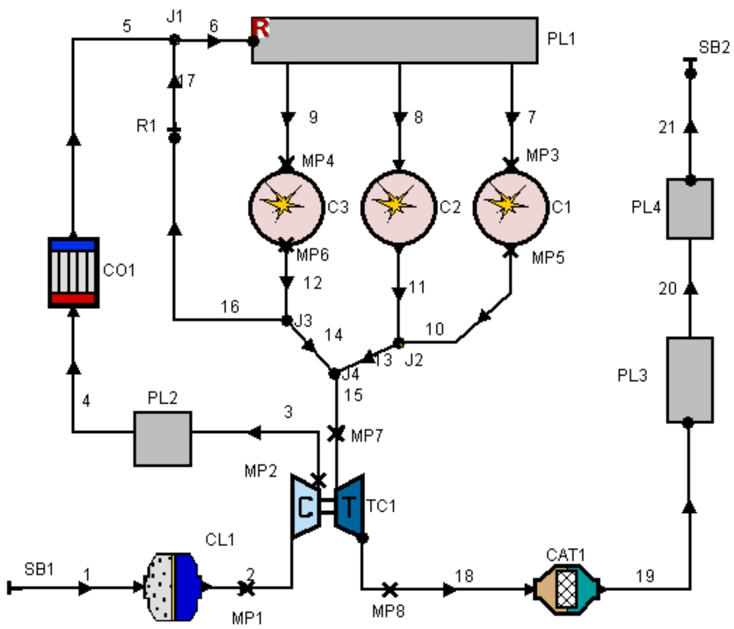


Figure 5—87: Model of a Turbocharged Diesel Engine with EGR

5.10.2. L4_1.4L_Industrial_Engine.bwf

This is a model of a natural aspirated diesel engine using the one zone model for combustion calculation based on Vibe burn rates.

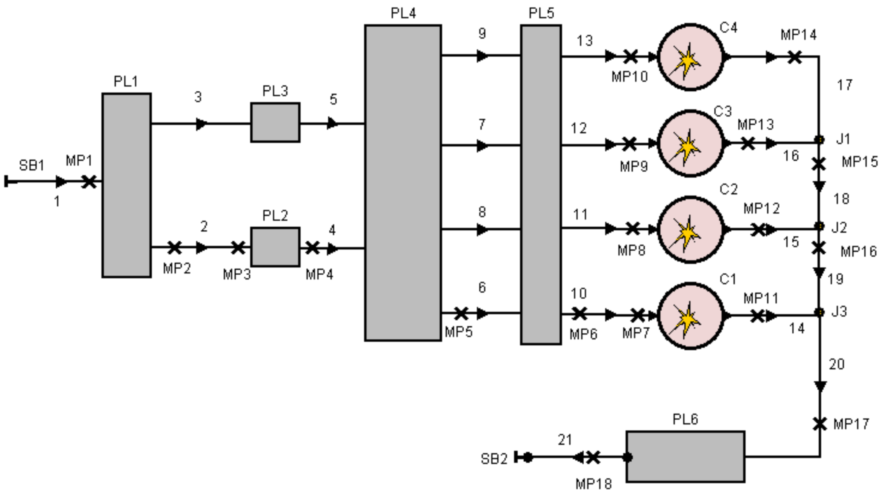


Figure 5—88: Model of an Industrial Diesel Engine

5.10.3. L4_1.6L_4V_GDI.bwf

This is a model of a GDI engine with a specified evaporation rate influencing the volumetric efficiency. The intake system is equipped with port control of engine turbulence. The effect of the turbulence is considered by its influence on combustion.

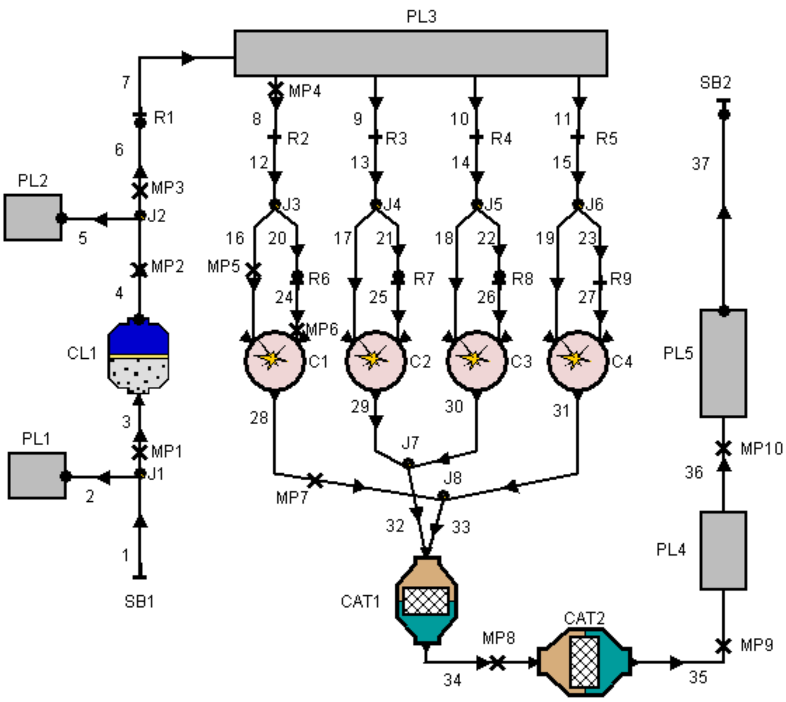


Figure 5—89: Model of a GDI Engine

5.10.4. L4_2.5L_DI_TCI_Diesel_Engine.bwf

This is a model of a turbo charged direct injecting Diesel engine, featuring a turbo charger for calculation with the simplified TC algorithm. The series model with an engine speed varying between 1000 rpm and 4000 rpm was created for the determination of the full load performance of the engine.

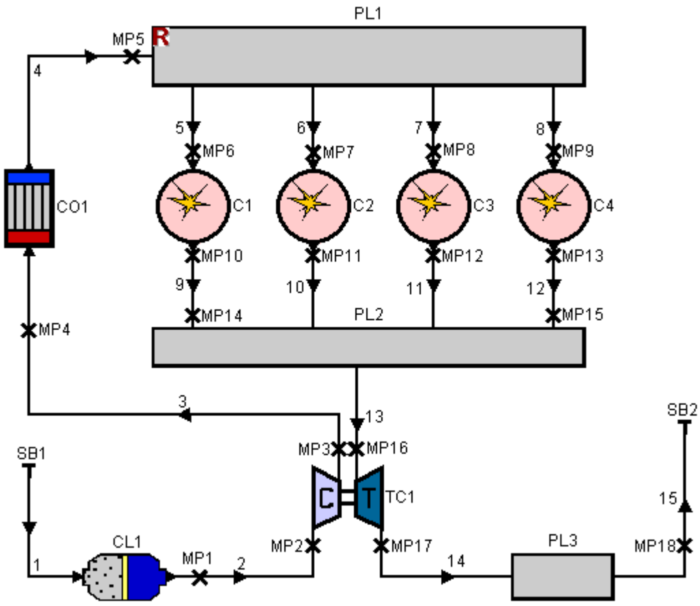


Figure 5—90: L4_2.5L_DI_TCI_Diesel_Engine.bwf File

5.10.5. L8_95L_SI_TCI_Gas_Engine.bwf

This is a model of a turbo charged, spark ignited, eight cylinder, four-stroke industrial engine with methane as the fuel. The engine speed is 1200 rpm. The bore and stroke of each cylinder are 245 mm and 250 mm, respectively and the compression ratio is 10.9.

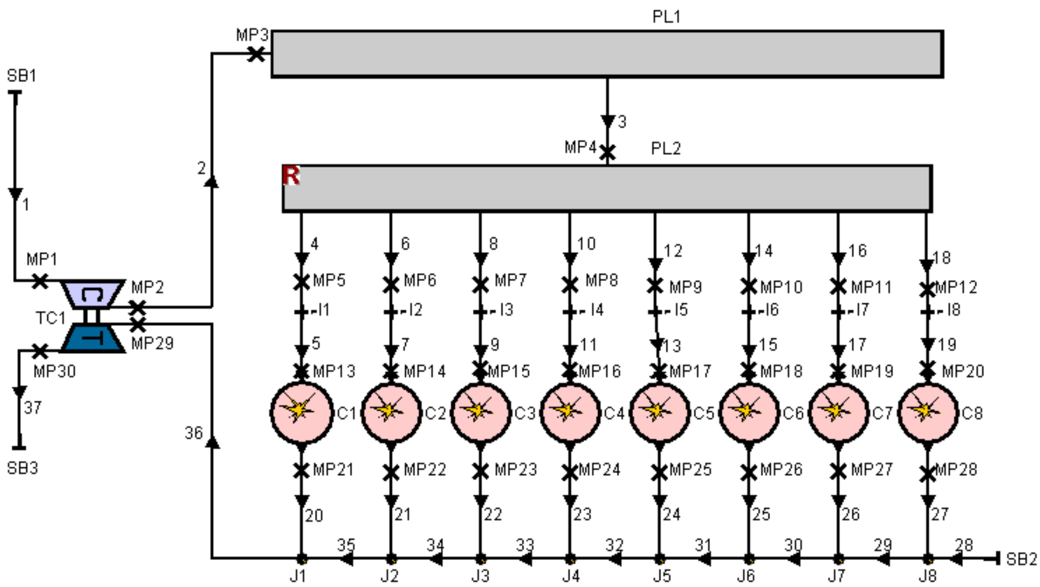


Figure 5—91: L8_95L_SI_TCI_Gas_Engine.bwf File

5.10.6. V6_2.7L_TCI_Bi-Turbo_Gasoline_Engine.bwf

This is a model of a 6 cylinder (V6) 2.7 litre TCI bi-turbo gasoline engine. This model of a passenger car engine features two separate turbo chargers (simplified model) and air coolers in the intake system, as well as two separate exhaust systems. The engine speed in this series calculation model varies between 1000 rpm and 6000 rpm.

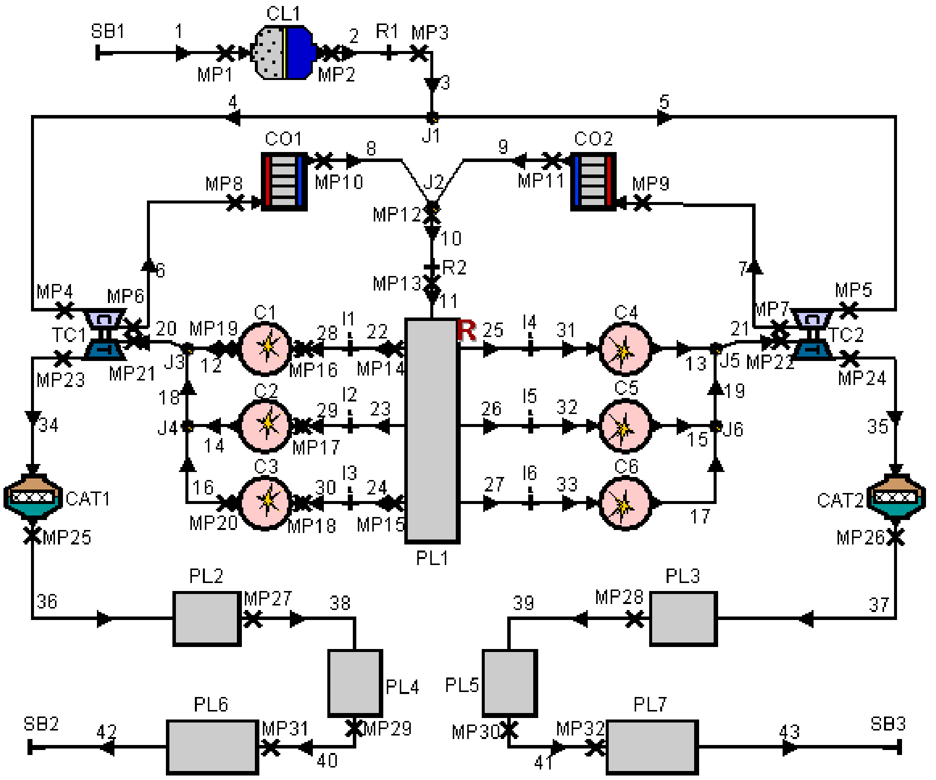


Figure 5—92: V6_2.7L_TCI_Bi-Turbo_Gasoline_Engine.bwf File

5.10.7. 6_Cylinder_2_Stroke_Diesel_TCI.bwf

This is a model of a 6 cylinder, 2-stroke diesel TCI engine. The bore is 900 mm, the stroke is 2100 mm and the engine speed is 120 rpm.

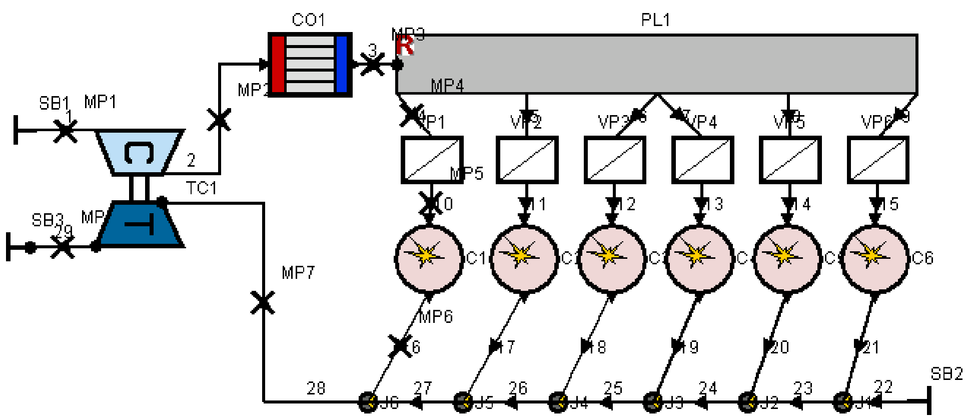


Figure 5—93: 6_Cylinder_2_Stroke_Diesel_TCI.bwf File

5.11. Spark Ignited

5.11.1. 2t1calc.bwf

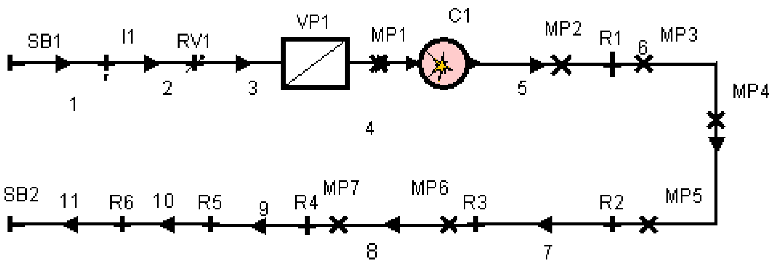


Figure 5—94: 2t1calc.bwf File

This model is a 2-stroke single cylinder racing engine. The crankcase is modeled with a variable plenum. After the gas dynamic active parts of the exhaust pipe up to restriction R4 the small diameter pipes 9 and 11 together with the larger diameter pipe 10 and restrictions R5 and R6 model a simple expansion chamber type silencer.

5.11.2. 4t1calc.bwf

Refer to the BOOST Primer Manual for the example description.

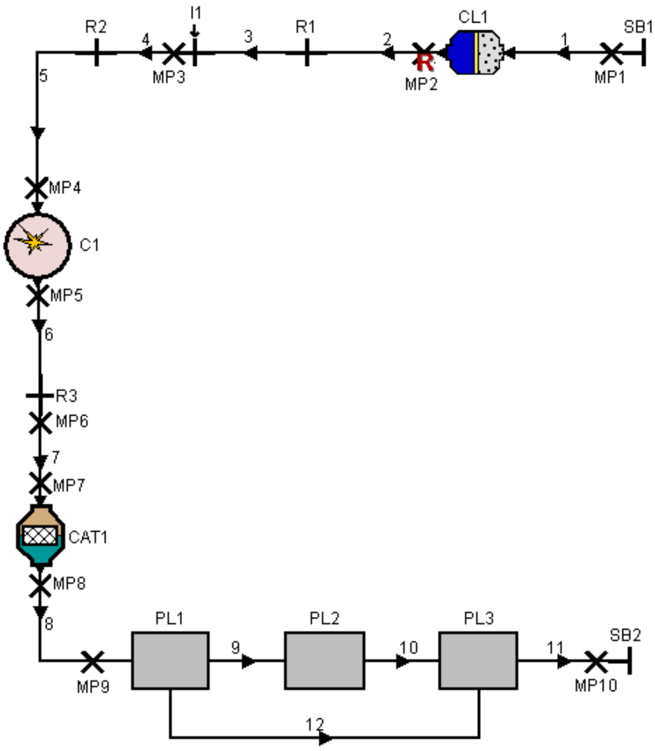


Figure 5—95: 4t1calc.bwf File

This is the model of a single cylinder motorcycle engine. It is a 4-stroke engine with a displacement of 500 ccm. The muffler has been modeled with its different chambers and the connecting pipes between them.

5.11.3. 4t1calc_species.bwf

The `4t1calc.bwf` case is converted to “General Species Transport”.

5.11.4. 4t1driver.bwf

An accelerated driving is performed on `4t1calc.bwf` using the BOOST driver model.

5.11.5. 4t1driver_chen_flynn_varwalltemp.bwf

The 4t1driver.bwf example is extended by:

- activated variable wall temperature feature for two pipes in the exhaust.
- Formula Interpreter and Monitor elements for setting the FMEP.

5.11.6. ottocalc.bwf

Refer to Chapter 2 for the example description.

The engine modeled is a 4-cylinder SI engine for passenger cars. The modeling of the intake manifold with junctions and pipes is of particular interest. This was necessary to reproduce the behavior of this engine component correctly.

5.11.7. ottocalc_species.bwf

The `ottocalc.bwf` case is converted to “General Species Transport”.

5.11.8. ottocalc_species_2zone.bwf

Based on the `ottocalc_species.bwf` case this model features a two-zone vibe combustion model in combination with detailed reaction chemistry in the burned zone.

5.11.9. ottocalc_E85_emissions.bwf

The `ottocalc_species_2zone.bwf` model extended by:

- multi-component fuel option for E85 (85% Ethanol, 15% Gasoline)
- consideration of the heat of evaporation in the injectors

5.11.10. ottocalc_transient_all_in.bwf

The `ottocalc_species_2zone.bwf` model extended by:

- transient operation
- general species transport
- intermittent injection with fuel puddling
- in-cylinder pollutant formation based on detailed reaction chemistry
- transient heat-up of cylinder walls
- air gap insulated exhaust port pipes (transient heat-up)

- reacting close coupled TWC model (warm-up and light-off)
- user defined friction model (Chen-Flynn) using the Formula Interpreter

5.11.11. ottoperf.bwf

A refined model of the exhaust system using perforated elements is added to `ottocalc.bwf`.

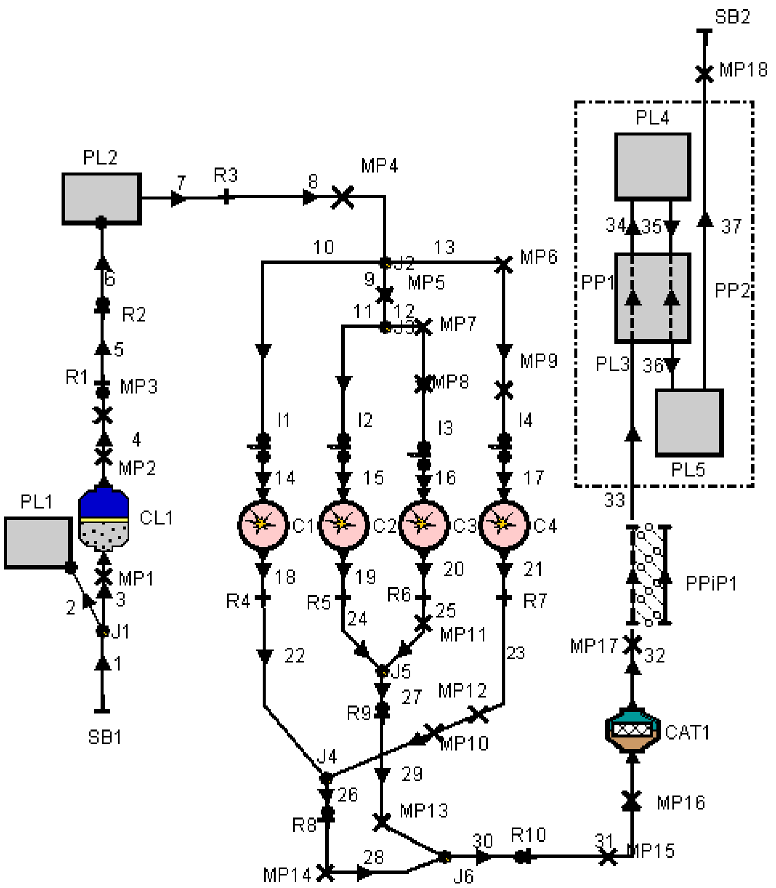


Figure 5—96: ottoperf.bwf File

5.11.12. ottoser.bwf

Refer to Chapter 3 for the example description.

The `ottoser.bwf` file contains the results of a series calculation with the engine speed as the main variation parameter.

5.12. Super Charged

5.12.1. first.bwf and scnd.bwf

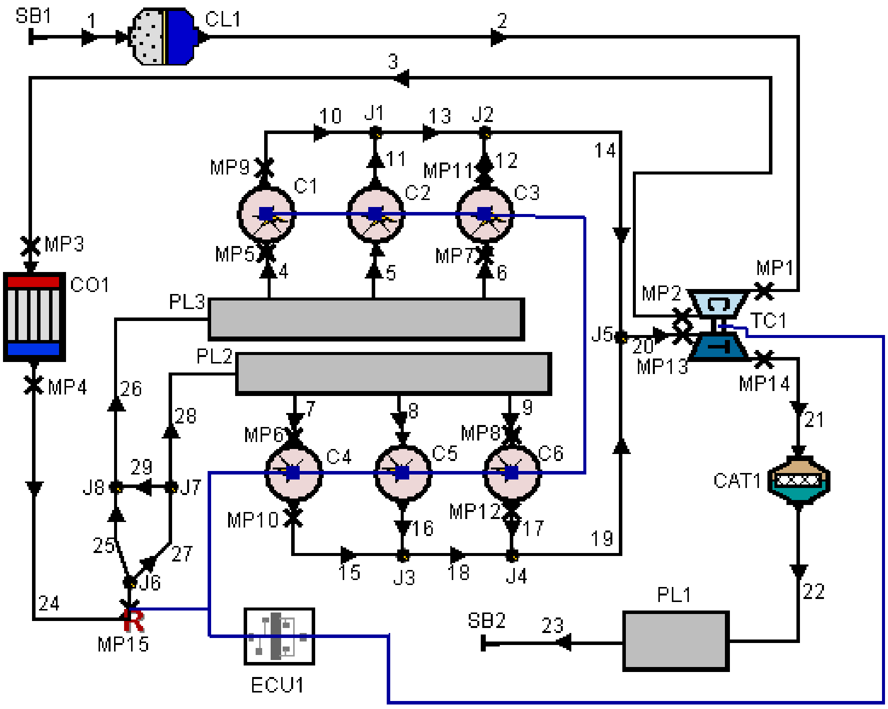


Figure 5—97: first.bwf and scnd.bwf Files

The engine simulated is a 6 cylinder HSDI diesel engine with a variable turbine geometry (VTG) turbocharger. The fuelling, injection timing and the vane position of the VTG are controlled by the engine control unit. The maximum fuelling is limited depending on the pressure at measuring point 15.

From the calculation with the `first.bwf` file, steady state conditions at approximately 1500 rpm, 0.7 bar load are obtained. The restart files are copied to `scnd.rs0` and `scnd.rs1`. With `scnd.bst` the calculation is restarted from these steady state conditions.

The engine accelerates at full load. If the control strategy is changed and the calculation is repeated with the same restart files, it ensures that the acceleration process always starts with the same initial conditions. Thus different control algorithms can be tested and compared easily.

5.12.2. first_power_turbine.bwf

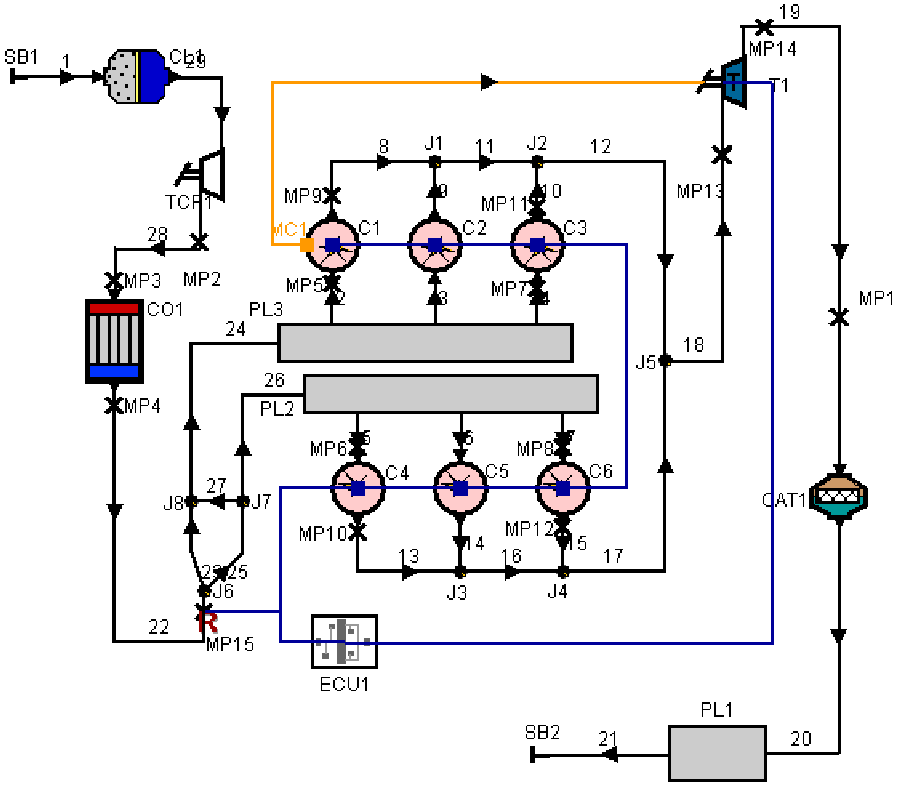


Figure 5—98: first_power_turbine.bwf

For the model first.bwf, the charging device is separated into a Simplified-Model Turbocompressor and a Full-Model Turbine which are mechanically connected to the crankshaft (due to compatibility reasons the Mechanical Connection of the Turbocompressor is not an own element, but still integrated in the Turbocompressor).

5.12.3. hyprex_hx95_test_bench.bwf

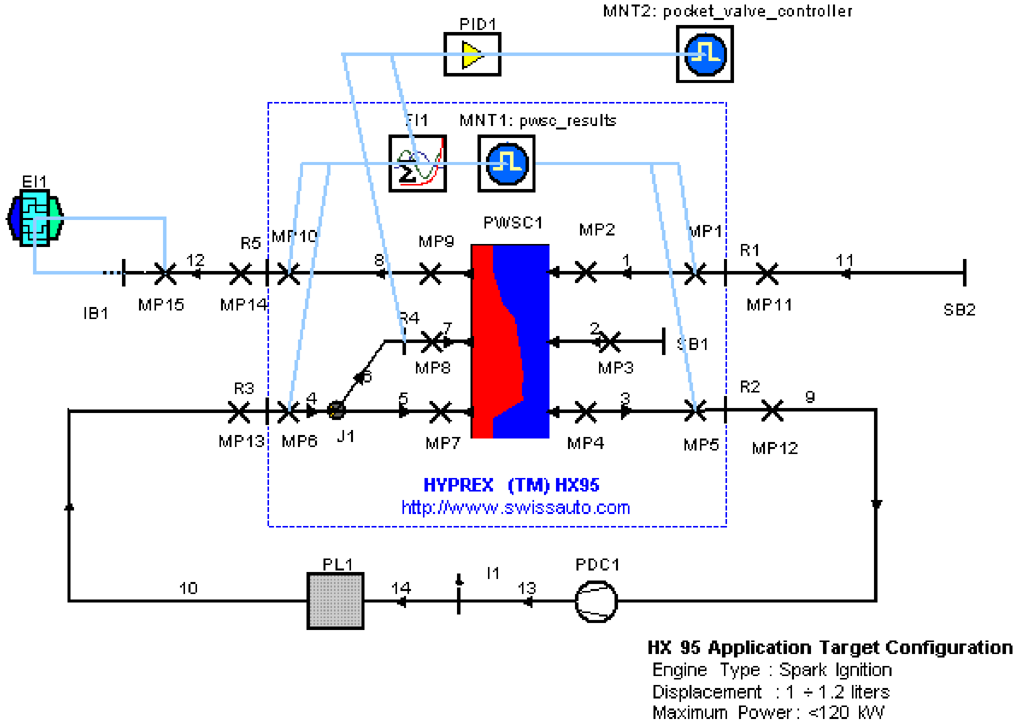


Figure 5—99: hyprex_hx95_test_bench.bwf

This model shows the Pressure Wave Supercharger Hyprex(TM) HX95 operated in a combustion chamber test bed environment. Plenum 1 is used in the Combustion Chamber mode, which is currently supported via User Defined Parameters only. According to the test bed operation PID 1 controls the Gas Pocket Valve (Restriction 1) to meet a target boost pressure. This configuration is set up for the model calibration of the Rotor Channel friction and heat transfer behavior.

Formula Interpreter FI 1 in combination with the Monitor MNT 1 is used to prepare a set of common PWSC Results.

The Target Engine for the Hyprex(TM) HX95 charging system is of the type spark ignition with a displacement in the range of 1 ÷ 1.2 liters and a maximum Power ≤ 120 kW.