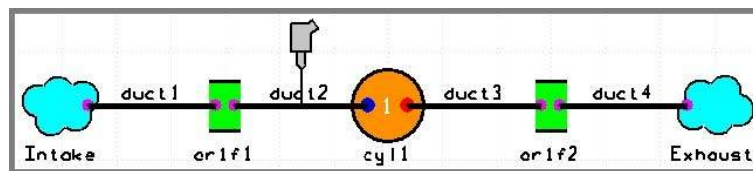


# SI Engine



Phase 1	Single Cylinder Model
Phase 2	The WAVE Solver and Time Plots in WavePost
Phase 3	Multi-Case, 4-Cylinder Model and Sweep Plots in WavePost
Phase 4	The Intake System and Animations in WavePost

## SI Tutorial, Phase 1 - Single-Cylinder Model



### 1. Starting WaveBuild, Setting General Parameters, and Creating a Simulation Title

#### 1.1 Starting WaveBuild

Open the WaveBuild GUI by following these steps:

Windows:

- ✱ From the Start menu select Programs -> Ricardo -> WAVE -> WaveBuild v7.0

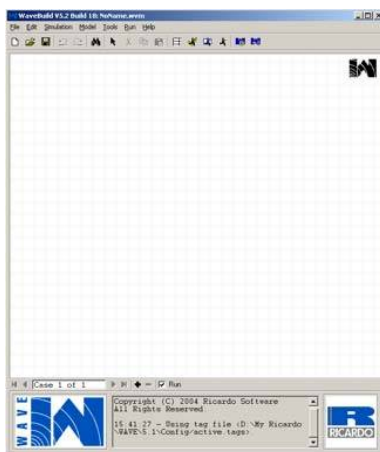


Figure 1: WaveBuild window at startup



Figure 2: Title bar when WaveBuild application first opens



Figure 3: Title bar with unsaved changes to the model  
(Note the asterisk (\*) located to the right of the file name)



Figure 4: Pull-down menu  
The toolbar contains shortcut buttons for frequently performed operations.

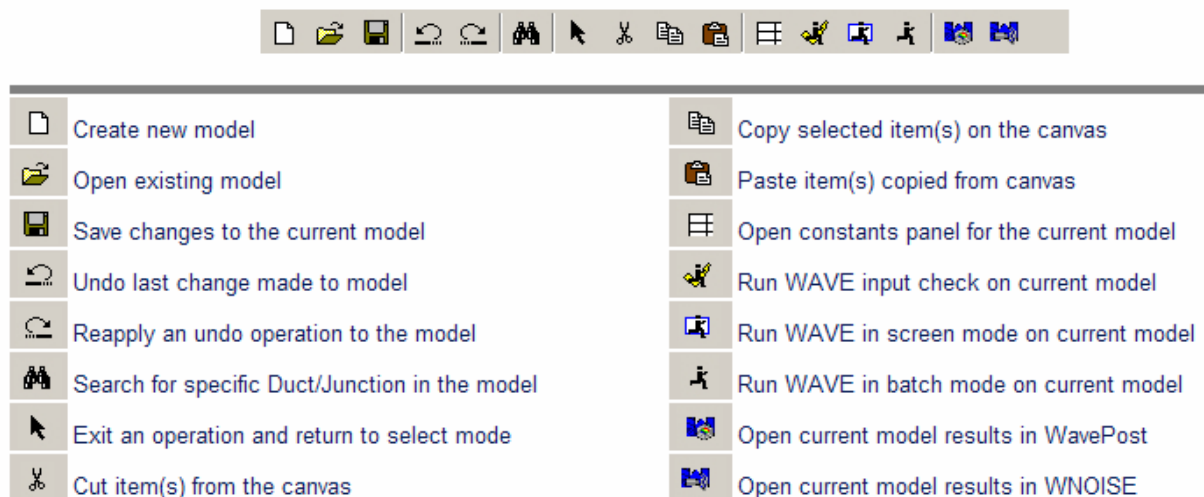
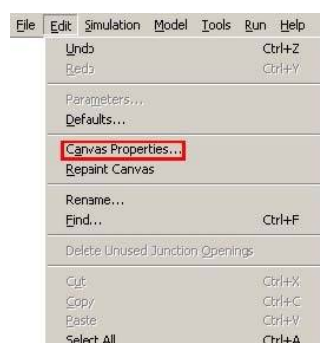


Figure 5: The WaveBuild canvas



## 1.2 Setting General Parameters

With any new model, the first step should always be to define the general parameters for the simulation, specifically the units system to be used for all data entry.

Open the **General Parameters** Panel by selecting General Parameters... under the **Simulation** pull-down menu.

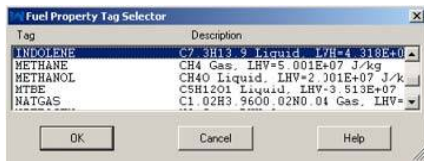


Figure 6: Selecting the INDOLENE tag

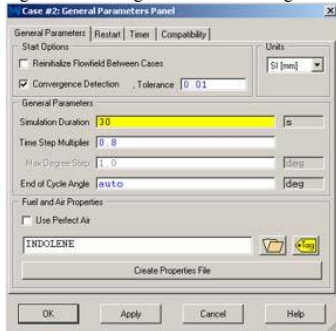


Figure 7: Completed General Parameters Panel

### 1.3 Creating a Simulation Title

Add a Title to the file by selecting Title... from the Simulation pull-down menu. The Title Panel will pop up for you to enter a text string that will serve as the simulation title. This title will be printed to the output file by WAVE at runtime so it is convenient to give the model a descriptive name for later reference. It can be up to 120 characters long and may include any alphanumeric characters or symbols with the exception of curly brackets, { }. Constants (discussed in [Step 4](#) of this tutorial) and operations on constants may be used and will be evaluated when placed inside curly brackets, { }. Pre-defined WAVE constants are convenient to use sometimes and include \$file, \$case, \$subcase, \$fullcase, \$version, and \$date. Also useful are user-defined constants that control important parameters such as engine speed and load. Type SI Tutorial, 4-Cylinder Gasoline Engine at {SPEED} rpm in the Title text field and click on the OK button to apply the title (the constant SPEED will be defined in [Step 4](#)). The simulation title appears centered across the top of the WaveBuild canvas and is fixed in this location. When finished, the WaveBuild canvas should appear as in Figure 8, right.

For Example: **SI Tutorial, 4-Cylinder Gasoline Engine at {SPEED} rpm**

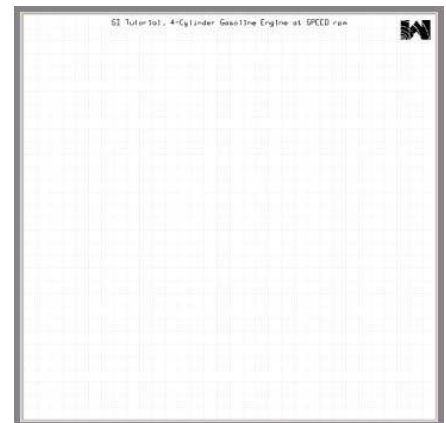


Figure 8: WaveBuild canvas with Simulation Title


### Save your model

Click on the Save button in the toolbar

# SI Tutorial, Phase 1 - Single-Cylinder Model

## 2. Building the Flow Network on the WaveBuild Canvas

### 2.1 Placing Required Junctions

Move the mouse over the Ambient junction button  and note the tooltip.

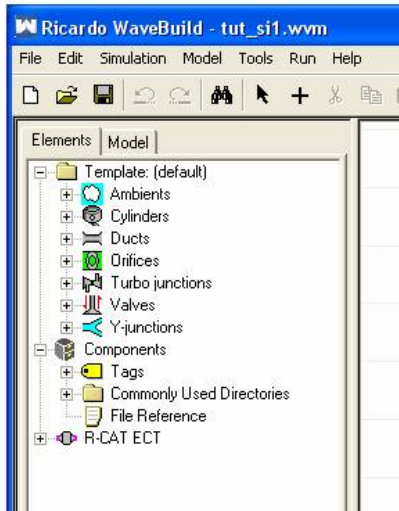


Figure 1: Location of the Junction Palette menu option

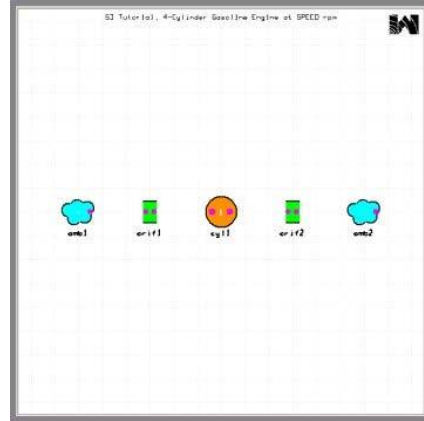


Figure 2: Junctions placed on the canvas

### 2.2 Connecting the Junctions with Ducts

Now that all of the junctions required for the single cylinder model have been placed on the canvas, ducts must be created to connect the junctions together. Using the left mouse button, click and drag from the pink connection point

on the leftmost ambient junction, labeled **amb1** in Figure 3, to the left connection point on the neighboring orifice junction, labeled **orif1** in Figure 3. This draws/creates a duct between the two junctions. The leftmost ambient will be the intake ambient and the rightmost will be the exhaust ambient. Connect the remaining junctions following the **Left to Right convention** (remember, the Left to Right convention doesn't necessarily mean Left to Right on the screen, it is merely a coincidence in this case!). When finished, the canvas should appear as in Figure 3.

**Note:** The ducts connecting the junctions appear as yellow lines. This is an indication that some geometric property of the ducts (i.e. diameter) has not been properly defined. Also, as you connect the two ports of an orifice junction, the icon of the orifice disappears because the ducts connected on both sides have default diameter values of 0 (zero) at this point. The orifice icon dynamically adjusts its appearance to reflect any step change in duct diameters. After you enter the duct properties in [Step 3](#), the orifice icon will "automagically" appear again and the ducts will turn black!

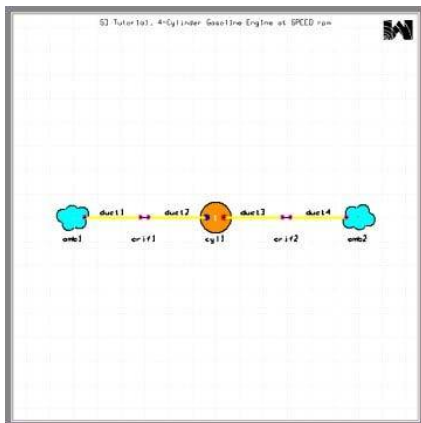


Figure 3: Ducts connecting Junctions on Canvas

### Save your model

Click on the Save button in the toolbar

# SI Tutorial, Phase 1 - Single-Cylinder Model

## 3. Defining Ambients, Ducts, and Orifices

In this step we will select all of the elements (ducts and junctions) on the WaveBuild canvas, and define their geometric values and initial/boundary conditions.

### 3.1 Defining Ambients

- ✱ Temperature and Pressure fields on the Ambient tab and all inputs on the Initial Fluid Composition tab are used to specify the fluid conditions of the atmosphere around the attached duct. The default values of 1.0 Bar and 300 K and a composition of 100% fresh air are suitable for this simulation and don't need to be changed. *In some simulations, the user may choose to dramatically change the temperature and pressure and/or adjust the composition to model a boundary that is either not atmospheric, like the outlet of a compressor or inlet of a turbine, or is not at sea level, such as an engine operating at high altitude.*
- ✱ The Diameter, Discharge Coefficient, and Acoustic End Correction fields are used to model the orifice created where the duct ends at the ambient. The Diameter field has a default value of AUTO to assume the same diameter as the attached duct (no restriction created). This value can never be larger than the diameter of the attached duct as it would have no physical meaning, however a value of 0 (zero) makes the ambient junction behave like a closed end to the attached duct (an end-cap). The Discharge Coefficient can be set to AUTO to have WAVE calculate this value internally during the simulation. This is only applied to flow going from the Ambient into the attached duct and should be a value between 0 and 1 if specified. The Acoustic End Correction is only used for acoustic simulations and is discussed in detail in the [Acoustics Manual](#).

For this simulation, the default values are all appropriate and we only need to change the name of the junction. Type Intake in the ID text field (name for the junction as displayed on the canvas and in the output files). When finished, the Ambient Panel should appear as in Figure 1, above. Do the same for the right-most ambient junction and type Exhaust in the ID text field.

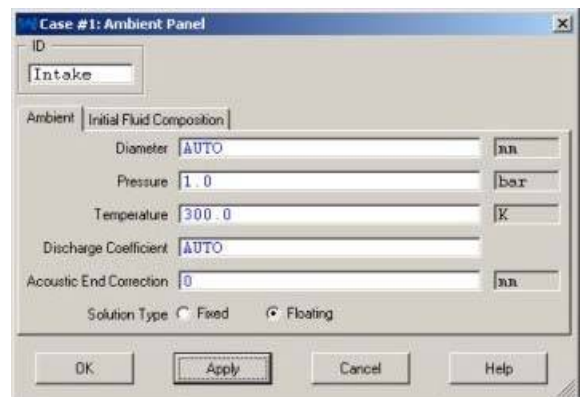


Figure 1: The Ambient Panel for the Intake

### 3.2 Defining Ducts

Next we will define the ducts. Assume that all the geometric data for the system has been measured and recorded and that a labeled sketch is provided as in Figure 2. Minimally, a duct is defined by Left and Right Diameters, Length, Discretization, and Initial Conditions. Double-click with the left mouse button on the duct labeled duct1 to open the Duct Panel.

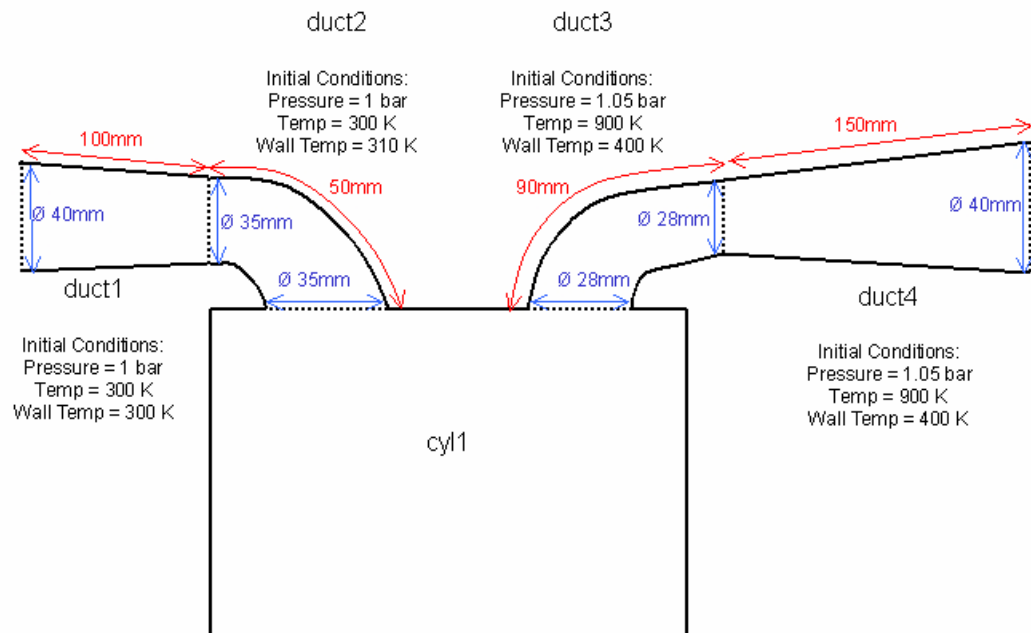


Figure 2: Single Cylinder Layout Sketch

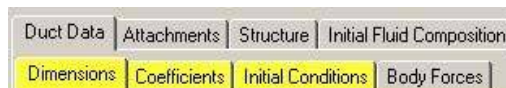


Figure 3: Required Duct Data Tabs

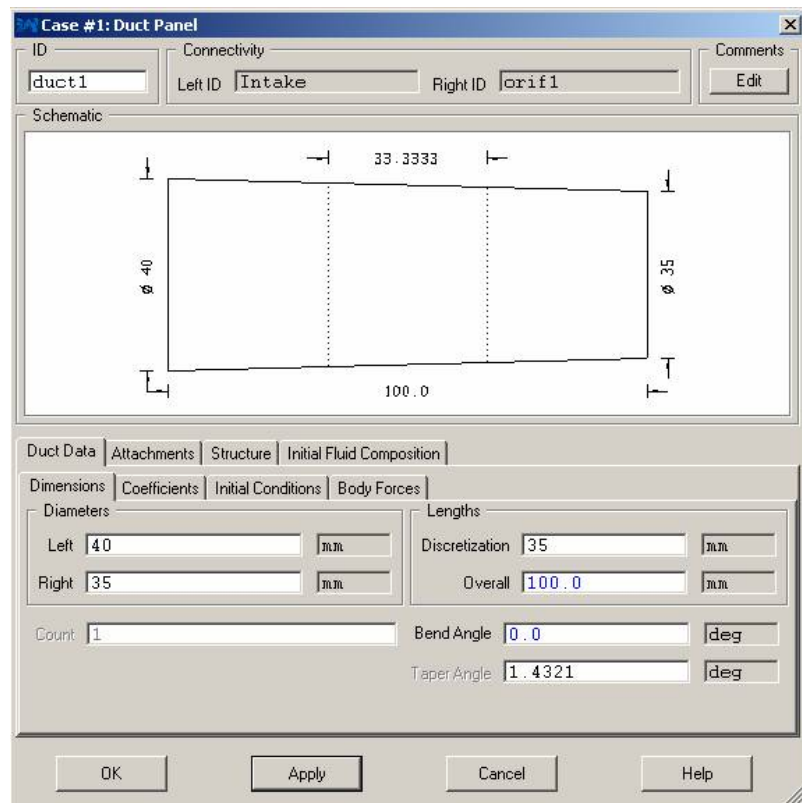


Figure 4: Duct Panel Dimensions Tab for duct1

**Case #1: Duct Panel**

ID:  Connectivity: Left ID:  Right ID:  Comments:

Schematic

Duct Data | Attachments | Structure | Initial Fluid Composition

Dimensions | Coefficients | Initial Conditions | Body Forces

Coefficients

Friction:

Heat Transfer:

Pressure Loss:

Discharge

Left:

Right:

Figure 5: Duct Panel Coefficients Tab for duct1

**Case #1: Duct Panel**

ID:  Connectivity: Left ID:  Right ID:  Comments:

Schematic

Duct Data | Attachments | Structure | Initial Fluid Composition

Dimensions | Coefficients | Initial Conditions | Body Forces

Initial Conditions

Pressure:

Temperature:

Wall Temperature:

Figure 6: Duct Panel Initial Conditions Tab for duct1

 Save your model

## 4. Defining the Engine

### 4.1 The Geometry Tab

To define the engine, open the Engine General Panel by selecting Engine... from the Model pull-down menu. The Engine General Panel consists of numerous tabs, each used to define the characteristics of the engine or the physical sub-models associated with the engine.

There are four primary tabs that are important for every engine. These are Geometry, Operating Parameters, Heat Transfer, and Combustion.



On the Geometry tab, under the Configuration section enter the relevant data for this engine as shown in Table 1.

<b>No. of Cylinders</b>	<b>1</b>
<b>Strokes per Cycle</b>	<b>4</b>
<b>Engine Type</b>	<b>Spark Ignition</b>
<b>Bore</b>	<b>78.1 mm</b>
<b>Stroke</b>	<b>82.0 mm</b>
<b>Connecting Rod Length</b>	<b>150.0 mm</b>
<b>Wrist Pin Offset</b>	<b>0.0 mm</b>
<b>Compression Ratio</b>	<b>10.0</b>

Table 1: Data to be entered in the Configuration fields

On the Geometry tab, under the Friction Correlation section enter the relevant data for this engine as shown in Table 2 to the right.

These coefficients are used in the Chen-Flynn friction correlation model. This model is used to calculate the FMEP (Friction Mean Effective Pressure) for the engine. When data is collected in the test cell, it can be plotted and correlated using the Chen-Flynn model so that FMEP may be calculated at non-tested engine speed/load conditions. The equation to calculate FMEP in WAVE is:

<b>ACF</b>	<b>0.35 bar</b>
<b>BCF</b>	<b>0.005</b>
<b>CCF</b>	<b>400 Pa/min*m</b>
<b>QCF</b>	<b>0.2 Pa/min<sup>2</sup>*m<sup>2</sup></b>

Table 2: Data to be entered in the Friction Correlation fields

$$\text{FMEP} = \text{ACF} + \text{BCF}(P_{\text{max}}) + \text{CCF}(\text{rpm} \cdot \text{stroke}/2) + \text{QCF}(\text{rpm} \cdot \text{stroke}/2)^2$$



**Case #1: Engine General Panel**

Geometry | Custom Geometry | Operating Parameters | Heat Transfer | Scavenge | Combustion | Conduction

**Configuration**

No. of Cylinders: 1

Strokes per Cycle: 4

Engine Type: Spark Ignition

Bore: 78.1 mm

Stroke: 82.0 mm

Connecting Rod Length: 150.0 mm

Wrist Pin Offset: 0.0 mm

Compression Ratio: 10.0

**Friction Correlation**

ACF: 0.35 bar

BCF: 0.005

CCF: 400 Pa\*min/m

QCF: 0.2 Pa\*min<sup>2</sup>/m<sup>2</sup>

**Firing Order and Relative TDC**

	1
Cylinder	1
TDC	0.0

**Printout Flag**

In-Cylinder State: 0

Port Conditions: 0

Imposed Piston Motion


OK Apply Cancel Help

Figure 1: The Engine General Panel Geometry tab

#### 4.2 The Operating Parameters Tab

Click the **Operating Parameters** tab to bring it to the front. In the **Engine Speed** text field, enter `{SPEED}` to denote the use of the SPEED constant (also used in the Simulation Title in [Step 1](#)). This will allow the simulated engine speed to change between cases once we add more cases in [Phase 3](#) of the tutorial. Note that the background of the text entry field turns yellow. This is to warn the user that the value is outside of the generally acceptable range of values for the **Engine Speed** field. Hover over the text `{SPEED}` and note the tooltip that pops up, stating that the constant `{SPEED}` is undefined ([Figure 2](#), below).

To define this constant, open the **Constants Panel** by clicking on the **Constants Panel** button in the toolbar

 or by selecting **Simulation -> Constants > Table...** from the pull-down menu. No constants have yet been defined so the **Constants Panel** should be blank. Type **SPEED** under the **Name** column and under the **Case 1** column, set a value of 6000. This will correspond to the `{SPEED}` value used in the Engine General Panel for **Engine Speed** in rpm. When completed, the **Constants Panel** should appear as in [Figure 3](#), below. Click **OK** to close the **Constants Panel** and save the setting.

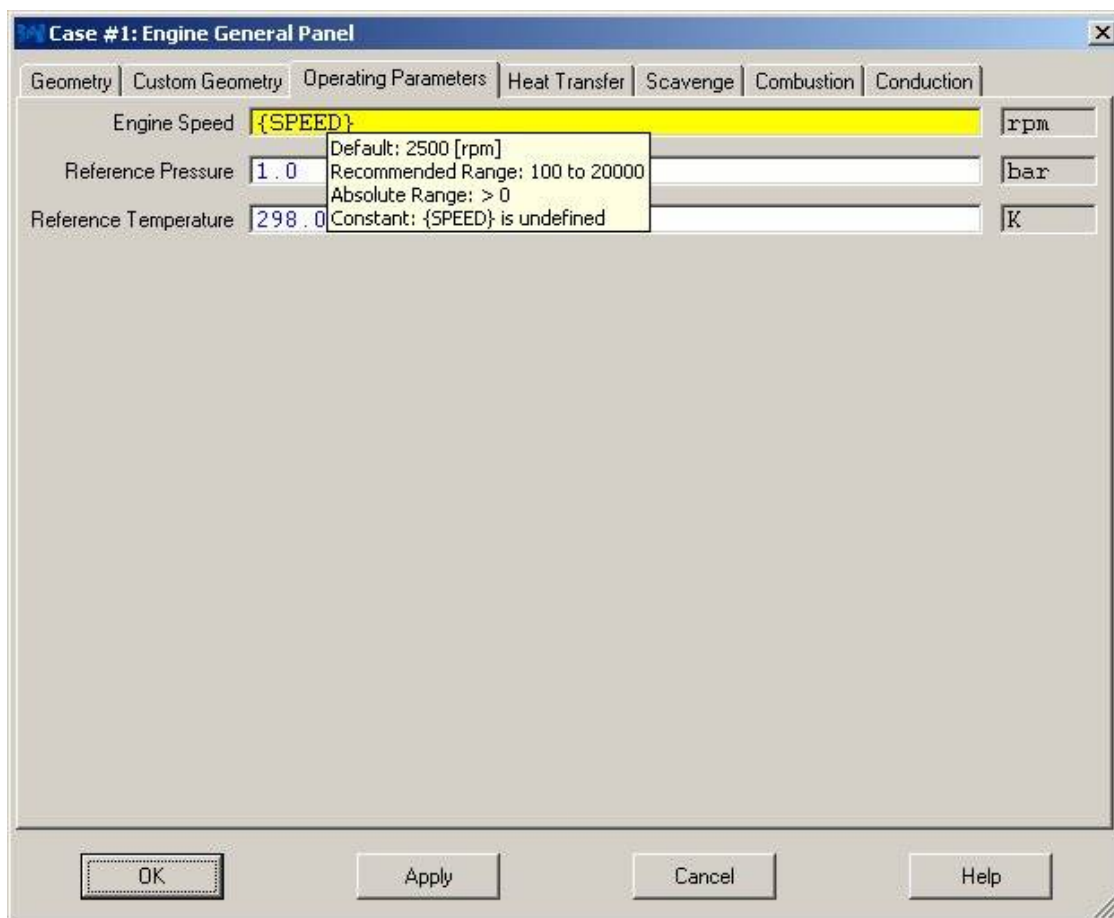


Figure 2: The Operating Parameters tab

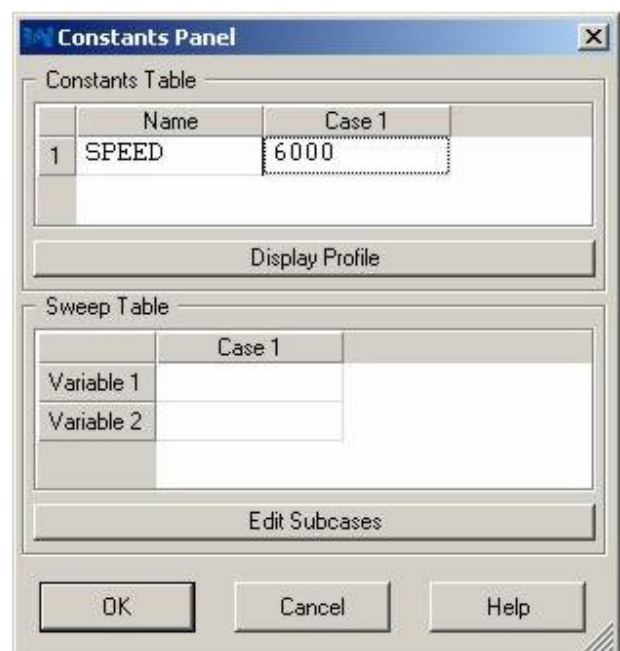


Figure 3: The Constants Panel

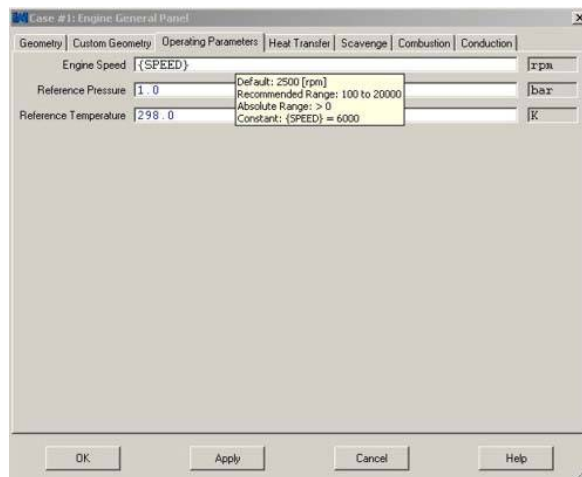


Figure 4: The Operating Parameters tab with SPEED defined

#### 4.3 The Heat Transfer Tab

Note: Clearance Height is in menu "Geometry"

Woschni Model	Original
Heat Transfer when Intake Valves are Open	1.0
Heat Transfer when Intake Valves are Closed	1.0
Piston Top Surface Area Multiplier	1.0 (flat-top piston)
Cylinder Head Surface Area Multiplier	1.6 (to make 7665 mm <sup>2</sup> )
Piston Top Temperature	520 K
Cylinder Head Temperature	520 K
Cylinder Liner Temperature	400 K
Clearance Height	2.0 mm
Swirl Ratio	0.0

Table 3: Data to be entered in the Heat Transfer Tab

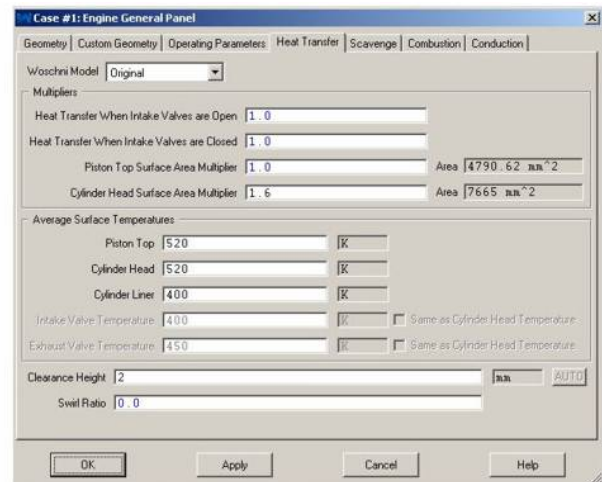


Figure 5: The Engine General Panel Heat Transfer tab

#### 4.4 The Combustion Tab

Finally, click the Combustion tab to bring it to the front. This panel separates the Primary Model choice (required) from the Secondary Model choice (optional, layered on top of the Primary Model) and Emissions Models. The general engine cylinder in WAVE simplifies combustion. It does not use a predictive combustion model but simply models the heat release caused by combustion vs. time. Since we are modeling a spark ignition engine and selected the Spark Ignition option on the Geometry tab, we have two combustion models available to use

- the SI Wiebe and Profile models.

The Profile model is used when Heat Release vs. Crank Angle data is available directly for every speed/load point to be tested by the model. This data can be directly entered into WAVE as a table with a combustion start time and efficiency.

More widely used, however, the SI Wiebe model simply uses an S-curve function that represents the cumulative heat-release in the cylinder. The first derivative of this function is the rate of heat release. The SI Wiebe model is very commonly used and represents experimentally observed combustion heat release quite well for most situations.

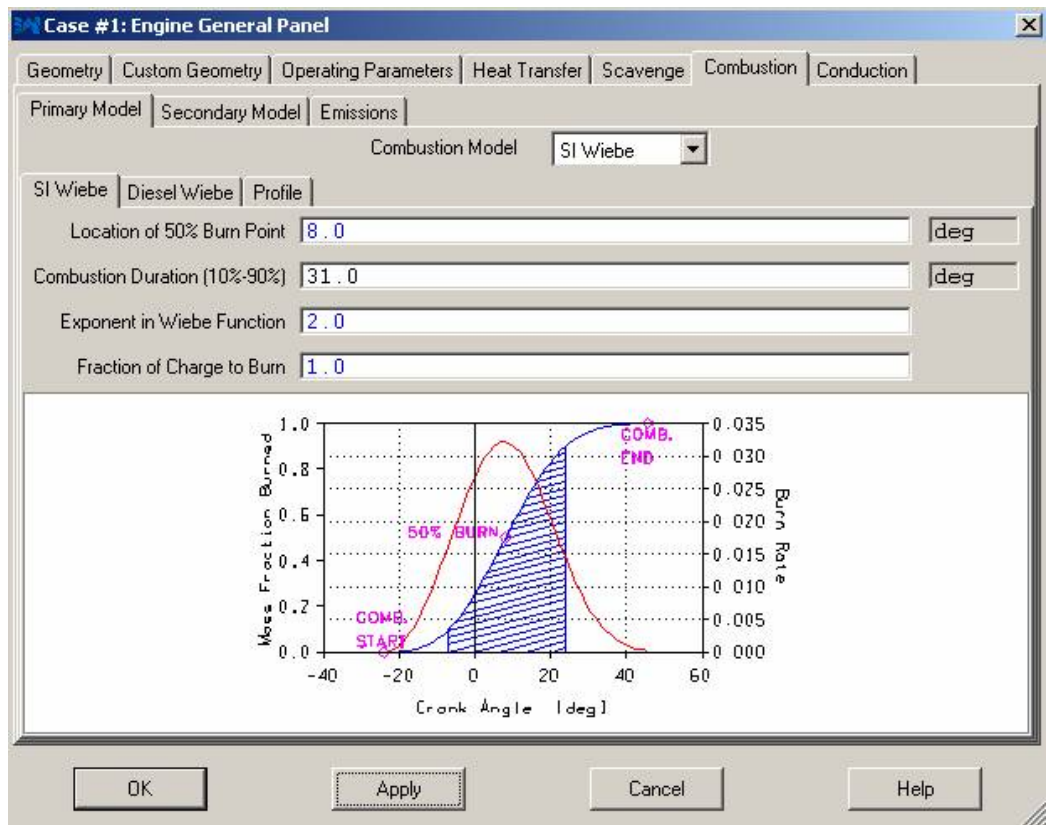


Figure 6: The Engine General Panel Combustion tab

Select the SI Wiebe option from the Combustion Model drop-down menu. Enter 8.0 (deg) for the Location of 50% Burn Point and 31.0 (deg) for the Combustion Duration (10-90%). Watch the plot actively update as these values are entered. The default value of 2.0 for the Exponent in Wiebe Function is appropriate for most cases. Change it and watch the shape of the burn curve change as well. For this example, 2.0 is an appropriate value and should be used. Fraction of Charge to Burn should be left at the default value of 1.0 as well. When completed, the Combustion tab should appear as in Figure 6.

## Save your model

## 5. Defining the Intake and Exhaust Valves

In this step we will define the intake and exhaust valves by specifying their lift behavior and flow restriction behavior.

### 5.1 Defining the Intake Valve Lift Behavior

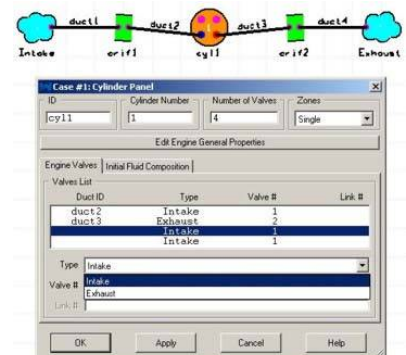
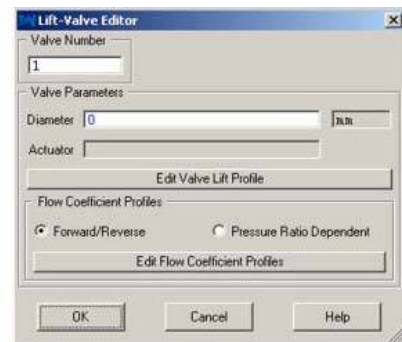
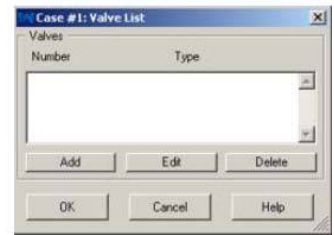


Figure 1: Adding Valve Connections via the Cylinder Panel

To define a valve, select **Valves...** from the **Model** pull-down menu. The Valve List panel will pop-up. As it is blank, it is clear that no valves have yet been defined. Click on the **Add** button to create a new valve. This will pop-up the Add Valve Panel where the user selects which type of valve to use. The standard valve on an engine cylinder is a Lift valve, selected by default. Click the OK button to accept the Lift type and the Lift-Valve Editor will automatically appear.



The first entry field under the **Valve Parameters** section of the panel is **Diameter**. This value is a reference diameter and is typically the inner-seat diameter (see Figure 2, right), but if the port-coefficient data has been provided in non-dimensionalized format, *whatever diameter was used to nondimensionalize the data should be entered here*. For this tutorial, type in a Diameter of 35 mm. Click on the **Edit Valve Lift Profile** button to open the Valve Lift Profile Editor.

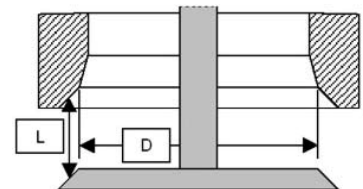
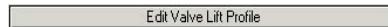


Figure 2: Valve Measurements typically used

In the Valve Lift Profile Editor, data must be entered for the behavior of the valve. This behavior is described as the lift of the valve vs. time (time is entered as cam or crank angle degrees). There are numerous options for entering this data including:

- \* Manually entering data into the array
- \* Copying and Pasting an array from MS Excel (on PC platform)
- \* Reading in a pre-formatted external file
- \* Using a tag to alias a pre-formatted external file

For this tutorial the data has already been provided in a pre-formatted external file that is aliased in the default active.tags file. To select this file, click on the tag button



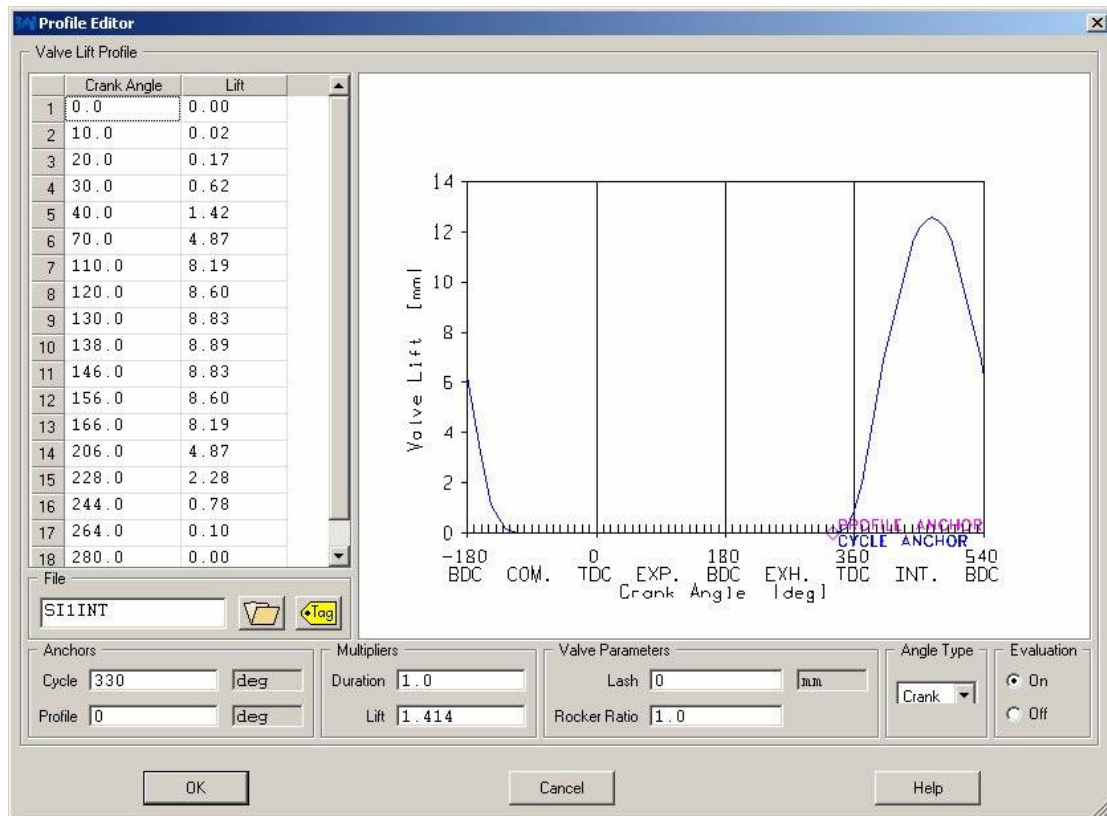
and select the SIINT item. Notice that the array fills automatically by reading the contents of the file aliased in the active.tags file and a lift vs. crank angle curve is now plotted on the screen.

For this tutorial, enter the following information for valve #1, the intake valve as shown below in Table 1.

This aligns the 0 degree point in the array data (in Crank Angle degrees) with the 330 degree point in the engine cycle, shifting the valve event over the labeled intake stroke in the plot. It also multiplies all of the lift values by 1.414. When finished, the Valve Lift Profile Editor for valve #1 should appear as in Figure 3, below.

INTAKE VALVE	
Diameter	35 mm
Lift Profile	SI1INT tag
Cycle Anchor	330 deg
Profile Anchor	0 deg
Duration Multiplier	1.0
Lift Multiplier	1.414
Lash	0
Rocker Ratio	1.0
Angle Type	Crank
Coefficient Profile	CFTYP tag

**Table 1:** Intake Valve Data



**Figure 3:** Valve Profile for the Intake Valve


Click the OK button to save these setting and close the Valve Lift Profile Editor.

## 5.2 Defining Coefficients (Flow vs. Discharge)

### Edit Flow Coefficient Profiles

Similar to the Valve Lift Profile Editor, data must be entered for the coefficients. This behavior is described as values for both the forward and reverse flowing direction (forward implies into the cylinder, reverse implies out of the cylinder) vs. the lift of the valve (non-dimensionalized by dividing the lift by the reference diameter). Again, there are numerous options for entering this data including:

- \* Manually entering data into the array
- \* Copying and Pasting an array from MS Excel (on PC platform)
- \* Reading in a pre-formatted external file
- \* Using a tag to alias a pre-formatted external file

For this tutorial the data has been provided already in a pre-formatted file, click on the tag button  and select the **CFTYP** option. Notice that the array fills automatically by reading the contents of the file aliased in the active.tags file and a coefficient profile appears in the plot on the right-hand side of the panel, see Figure 4.

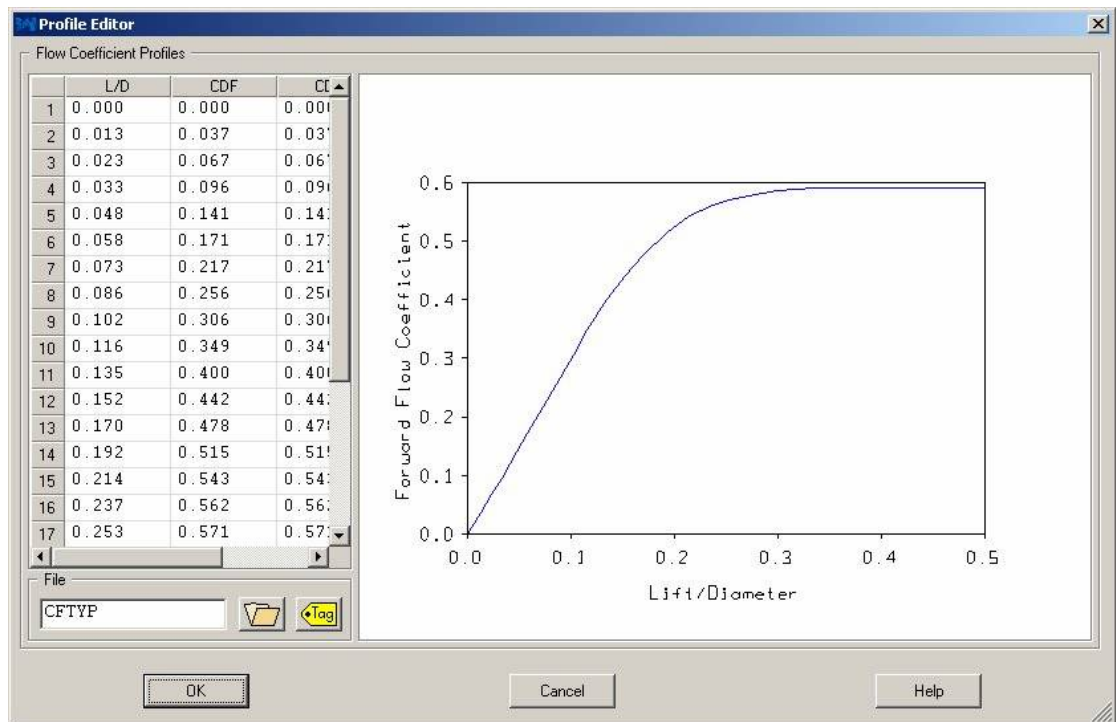


Figure 4: Typical Flow Coefficient Profile

Click the OK button to save these setting and close the Profile Editor.

## 5.3 Defining the Exhaust Valve

Click the Add button again to **create** a second valve that will be used to model the exhaust valve.

Follow the same steps as above but use the following information for the lift profile (the coefficients can be the same as the Intake Valve). When finished, the Valve Lift Profile Editor for valve #2 (the exhaust valve) should appear as in **Figure 5**, below.



EXHAUST VALVE	
Diameter	28 mm
Lift Profile	SI1EXH tag
Cycle Anchor	105 deg
Profile Anchor	0 deg
Duration Multiplier	1.0
Lift Multiplier	1.0
Lash	0
Rocker Ratio	1.0
Angle Type	Crank
Coefficient Profile	CFTYP tag

Table 2: Exhaust Values

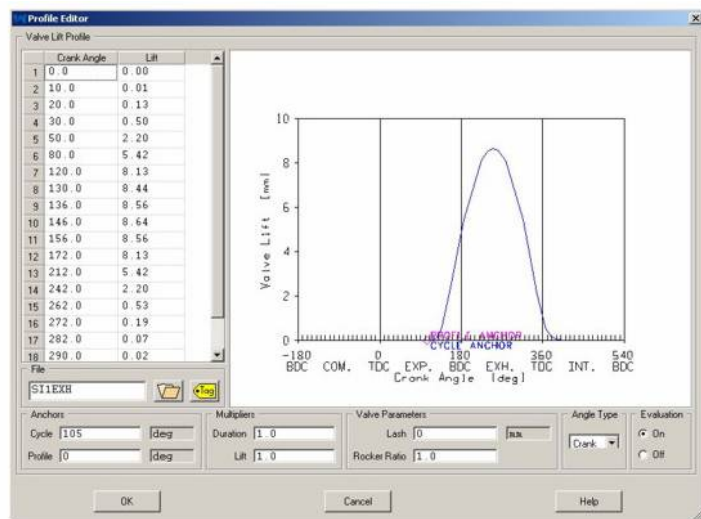


Figure 5: Profile Editor for Valve #2 (the exhaust valve)

With both the intake valve (valve #1) and exhaust valve (valve #2) defined, the Valves List should appear as in Figure 6. Click the OK button to save these setting and close the Valves List panel.



Figure 6: The Valves List

**Save your model**

## 6. Adding the Fuel Injector

### 6.1 Defining the Global Injector

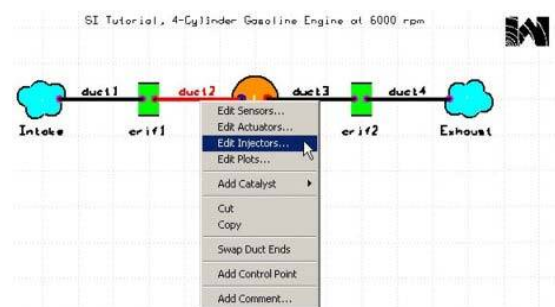


Figure 1: Adding an Injector

Right-click on the **duct2**, where the injector will be located. From the context menu that pops-up, select **Edit Injectors...** (see Figure 1). To globally define an injector, click on the **Edit Duct Injector Types** button.



This will open the Duct Injector Type Editor as in Figure 2.



**Case #1: Duct Injector Type Editor**

Injector Types

Type Number	Characteristic
1	prop

Add Delete

Properties

Type Number: 1 Characteristic: Proportional

Properties Composition Profiles

Mixture Temperature: 330 K

Flow Rate When On: 0 kg/hr

Nozzle Diameter: 0.2 mm

Mean Fuel Drop Diameter: auto um

Sac Volume: 0 mm<sup>3</sup>

Liquid Fraction Evaporated After Injection: 0.3

Spray Spread Angle: 40 deg

Initial Fuel Injection Velocity: Auto

Velocity: 30 m/s

OK Cancel Help

Figure 2: Completed Duct Injector Type Editor

Click on the **Composition** tab, where the total composition of the fuel before injection can be specified. If the aforementioned charge cooling effect is undesirable, then the vapor portion can be specified here. For this tutorial, the default of **1.0 for Liquid Fuel** is suitable and can be left as is (100% of the injected fuel is in liquid state, 20% of that vaporized when injected). When completed, the Duct Injector Type Editor should appear as in Figure 2. Click the OK button to save the data for this global injector and the panel will close.

## 6.2 Placing the Local Injector

Now that the global injector has been defined, we can add the injector to duct2. On the **Duct Injector Editor**, click the **Add** button to place the injector on duct2. The injector should now appear in the list of attached injectors. Click on the **Edit** button to assign the behavior to the injector that is specific to this location. This will open the Injector Editor panel.

For a Proportional type injector that is placed on a duct, all that needs to be defined is the targeted air-fuel ratio and the placement of the injector within the duct. WAVE requires a *fuel-air* ratio but most frequently *air-fuel* ratio information is provided. This is easily overcome by using WAVE's capability to perform simple mathematical operations on constants. We will define a constant named A\_F and enter air-fuel ratio data in the Constants Panel, but in the text field for Fuel/Air Ratio, type **{1/A\_F}**. This will automatically convert the air-fuel ratio to a fuel-air ratio as required.

The Injector Editor dialog box contains the following fields and controls:

- Injector Number:** Text field with value "1".
- Duct Number:** Text field with value "duct 2".
- Injector Type:** Dropdown menu with value "1 prop".
- Operating Point / Position:** Tabbed interface with "Position" selected.
- Injection Rate:** Text field with value "0" and unit dropdown "kg/hr".
- Start of Injection:** Text field with value "0" and unit dropdown "deg".
- Injected Mass:** Text field with value "0.0" and unit dropdown "mg".
- Duration of Injection:** Text field with value "0.0" and unit dropdown "ms".
- Fuel/Air Ratio:** Text field with value "{1/A\_F}".
- Synchronize to Cylinder:** Dropdown menu with value "None".
- Buttons:** OK, Apply, Cancel, and Help.

On the position tab, type **25 (mm)** into the Distance from Left End text field to move the injector to the middle of the duct (alternatively, click and drag the injector with the middle mouse button). When completed, the Injector Editor panel should appear as in Figure 3 below. Click the OK button to close the Injector Editor and save the data (when prompted to add the A\_F constant to the Constants Panel, select No) . Click the OK button again to close the Duct Injector Editor panel and save the data.

The completed Injector Editor dialog box shows the "Position" tab with a visual representation of the injector's location. A horizontal scale from 0.0 to 90 is shown, with an injector icon positioned at 25. The "Distance From Left End" text field contains the value "25 mm".

Figure 3: Completed Injector Editor

An injector icon should now appear, branching off of duct2 on the WaveBuild canvas (see Figure 5). All that remains is to add the A\_F constant to the Constants Panel. Open the Constants Panel , right-click on the number 1, and select **Insert Row After** from the menu that pops up (see Figure 4). In the newly created row, type the constant name **A\_F** and under the **Case 1** column, enter the value **14.7** (approximately stoichiometric for the INDOLINE fuel that this simulation is using). Click the OK button to close the Constants Panel.

**The single-cylinder model is complete!**

	Name	Units	Case 1	+
Status			Run	Run
1	A_F		14.7	
2	SPEED	rpm	6000	
+				

**Save your model**


## SI Tutorial, Phase 2 - The WAVE Solver and Time Plots in WavePost

## 1. Running a WAVE Input Check

When finished building a model, the first thing to do is check to make sure all of the entered settings are acceptable for the WAVE solver to process. Although WaveBuild itself has many error checking mechanisms, the Input Check validates *all* settings to ensure that the WAVE solver will run.

## 1.1 Starting the Input Check

All of our geometry has been defined and all of the initial conditions and boundary conditions have been set. We are ready to run our simulation but first we will check

to make sure that everything we have entered is acceptable to WAVE. Click on the Run Input Check button  on the toolbar. This will launch a shell window in which the WAVE solver will run an Input Check (see Figure 1 to the right). The Input Check consists of the solver internally assembling the network and initializing the gas state of every element within the model. If it can successfully perform these tasks, the model is ready to run a full analysis in the solver.

When the Input Check is successful, the last item printed is the simulation title.

[illegible]

Figure 1: The Input Check Shell

## 1.2 Interpreting WAVE's Output

Standard output from the simulation is printed to the shell but is also printed to a file having the same prefix as the model file but with a .out extension. This .out file will be created in the same directory as the .wvm file.

The .out file contains all of the output printed to the screen during the input check as well as some other useful information. Close the Input Check shell window and open the .out file that has been created using a text-editor.

The first item printed to the .out file is the header that states the version of the WAVE solver being used. Following that is the data as entered in the model file (the input is echoed to the output for reference). And finally, the information that is printed to the shell window during the input check is also printed at the end of the .out file.

Note that the constants used (pre-defined and user-defined) are all evaluated to check for validity.

WAVE prints three types of messages to user on the screen with which we should be familiar.

1. Simple informational messages begin on a line that starts with **I\*\*\***.
2. Warning messages, to warn the user that something may or may not be of concern begin with **W\*\*\***.
3. Finally, failure messages, which causes WAVE to stop the simulation, begin with **F\*\*\***. Failure messages in WAVE tend to be rather descriptive and can be of great help in debugging problems.

## 2. Requesting Time Plots

In this step we will add requests for plots of specific data to our existing model. Time Plots are plots of results over the course of a single engine cycle (the last engine cycle in multi-cycle simulations). Time Plots should be requested whenever the user is aware of specific data they are interested in analyzing.

## 2.1 Duct Time Plots

Suppose we are interested in observing the pressure and temperature at the mid-point of the intake port during the engine cycle. We can request Time Plots at the intake port (duct2) and WAVE will automatically create these plots at the end of the simulation. **Right-click on duct2** and select the **Edit Plots...** option from the popup menu (see Figure 1). The Duct Plot Panel will open with no plots yet defined at this location. Click on the **Create Plot** button to open the Duct Plot List.



The list of plots available at this location is displayed (this list is context-sensitive and generated based on the canvas item selected). Click on the **201 PRESSURE** plot and then, holding the Shift button to multiple-select, click on the **202 TEMPERATURE** plot (most valid duct plots are in the 2xx range). Click on the OK button to close the Duct Plot List and add these plots to the Existing Plots list in the Duct Plot Panel.

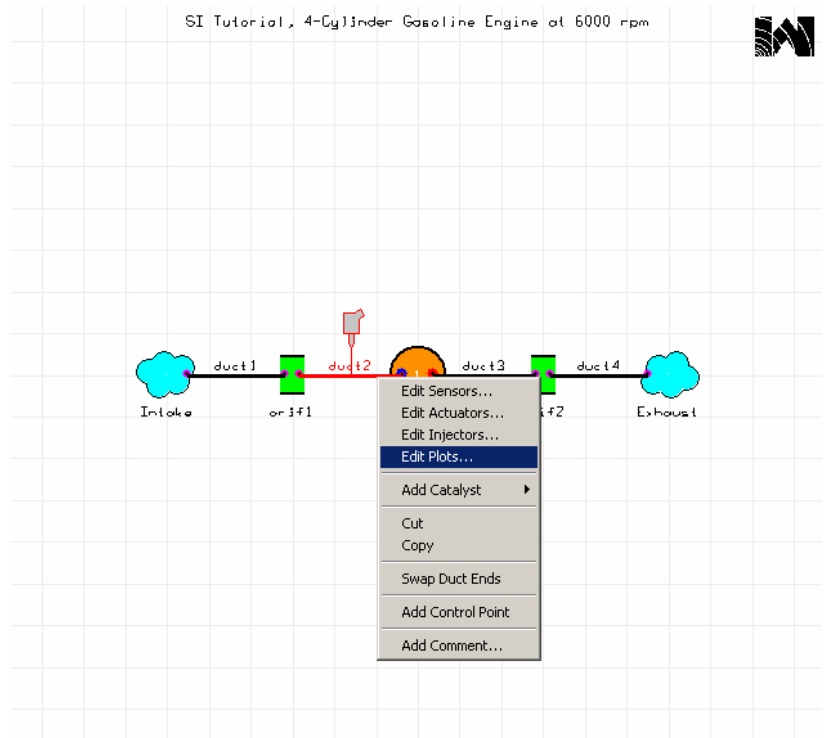


Figure 1: Adding Plot to duct2

When complete, the Duct Plot Panel should appear as in Figure 2. Click on the OK button to close the Duct Plot Panel. Note the plot icon now hanging off of duct2. To edit plots at this location again, simply double-click on this icon and the Duct Plot Panel will open.



Figure 2: Duct Plot Panel with plots at duct2

## 2.2 Junction Time Plots

Suppose we desired to examine the combustion performance in the power cylinder during the engine cycle. We may want to create a P-V (Pressure vs. Volume) plot. Right-click on the **cyl1** junction and select the **Edit Plots...** option to open the Junction Plot Panel. The duct plots created above will be listed in the Existing Plots list. Click on the **Create Plot** button to open the Junction Plot List.



Note that there are many more plots in the list than in the Duct Plot List (engine cylinder plots are typically in the 1xx range). Click on the **111 LINEAR P-V PLOT** and click the **OK** button to close the list and add the plot to the Existing Plots list. Also note that an asterisk appears to the right of the plot type for any plot with the currently selected canvas item as a location.

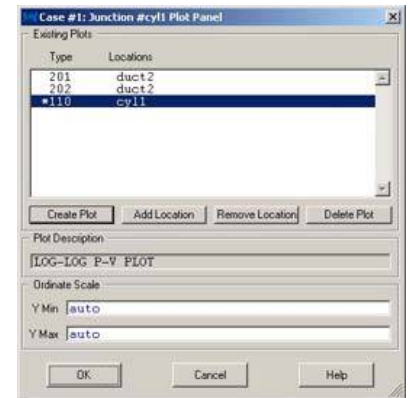


Figure 3: Junction Plot Panel with plot at cyl1

No location within the junction needs to be specified for the plot as there is only one calculation point in a junction. When completed, the Junction Plot Panel should appear as in **Figure 3**, right. Click the **OK** button to close the Junction Plot Panel.

### 2.3 System Time Plots

Although there is only one cylinder in our engine, we will eventually be creating a 4-cylinder engine. To observe the behavior of the engine itself (system of cylinders grouped together), click on the **Simulation** pull-down menu and select the **Time Plot...** menu item. This will open the Time Plot Panel where plots can be created for the engine system, as well as sensors, actuators, and pins (to be discussed in more advanced tutorials).

The System Location Type should be active by default when the panel opens. Click on the **Create Plot** button to open the Time Plot List for the engine system.



A few engine system specific plots are available and even fewer are applicable to our system as modeled (system plots are typically in the 7xx range). Click on the **701 ENGINE TORQUE** plot and click the **OK** button to close the list.

Again, no location needs to be defined for system plots. When completed, the Time Plot Panel should appear as in **Figure 4**. Click the **OK** button to close the Time Plot Panel. Note, there is no display of the system plot on the canvas as there is not yet a canvas entity to which it can be attached.

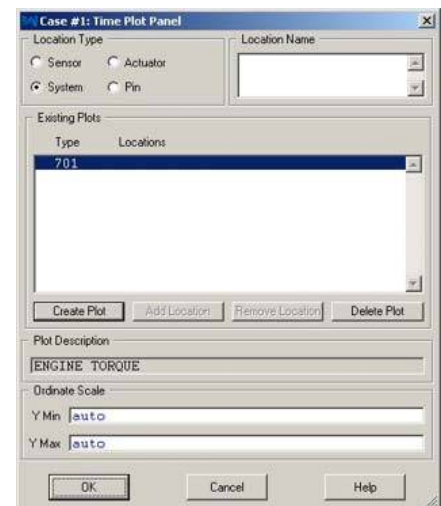


Figure 4: Time Plot Panel with system plot of Engine Torque

### 2.4 Multiple Plots Overlaid

Perhaps we wish to examine the pressure and temperature in the exhaust port and compare it to the conditions in the intake port. We could create separate plots for the duct representing the exhaust port (duct3), but it would be more useful if the data for both ports were on the same plot. **Right-click on duct3** and select the **Edit Plots...** option from the pop-up menu. Note that the only plots in the Existing Plots list are the pressure and temperature plots from duct2. This is because these plots are allowed at duct3

as well (a P-V plot is not sensible in a duct, nor is engine torque).

With plot **201 highlighted**, click on the **Add Location** button to plot the pressure at **duct3** on the same plot as **duct2**.

Add Location

Do the same for plot **202** to add **duct3** to the plot of temperature in duct2. Click on the **Use All Locations** button to request the plots at the center of both cells in duct3 (locations of 0.25 and 0.75).

When completed, the Duct Plot Panel should appear as in **Figure 5** to the right. Click the OK button to close the Duct Plot Panel. The model should appear as in **Figure 6** with all time plots added.

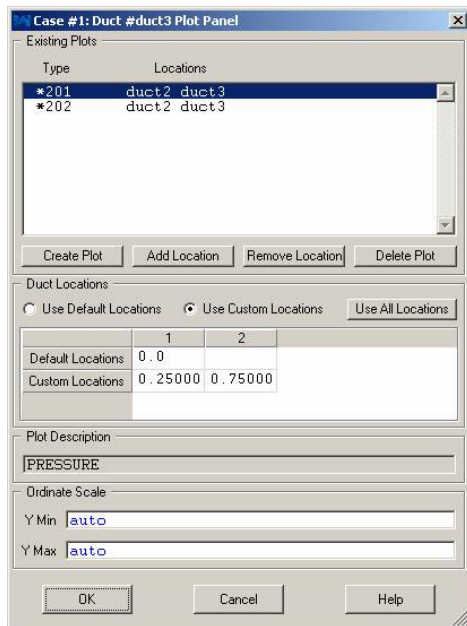


Figure 5: Time Plot Panel with system plot of Engine Torque

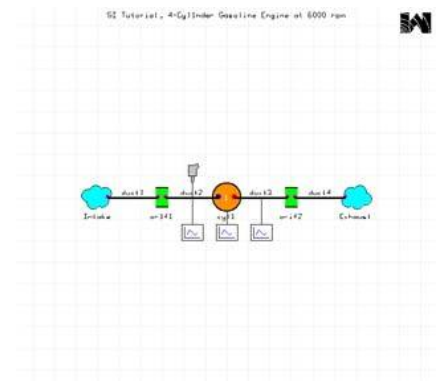


Figure 6: Model with Time Plots Added

**Save your model**

### 3. Requesting Post-Processing Datasets

In this step we will request that particular data that we may be interested in later be output for the *entire network*, as opposed to *certain locations* as in the case of Time Plots. This is more convenient if the user wishes to have the data (e.g. Pressure, Temperature, etc.) at numerous points throughout the network or is unsure, ahead of time, at which locations this data will be required for post-processing. It also allows the user to do more advanced post-processing to be discussed later in this tutorial.

Chapter sections Example Input File:

### 3.1 Basic Datasets

Suppose we are not sure, beyond the behavior in the intake and exhaust ports, which behaviors we will be interested in viewing results for at the end of the simulation. Later analysis might require extensive amounts of data that we didn't request plots for ahead of time! It may also be inconvenient to return to WaveBuild and request more plots and rerun the model, especially if the model takes a very long time to run! We can avoid some of this by requesting Post-Processing Datasets in advance.

Open the Postprocessing Output Panel by selecting the **Postprocessing Output...** menu item from the **Simulation** pull-down menu. We have yet to request any data to be output after the simulation is run through the WAVE solver, so the Requested Datasets list should be blank. By default, the Basic models are listed, which are datasets grouped together that are always available, no matter which junctions or physical models exist in the simulation. To request a specific dataset that we would like written to the simulation output file, highlight one by clicking on it and then click on the single arrow button pointing to the right to add it to the list of Requested Datasets (datasets can be multiple-selected by holding down the Shift key to highlight a span of datasets or the Ctrl key to highlight multiple, individual datasets). Clicking on the double-arrow button will move all datasets from a Model grouping in the the Requested Datasets list.

For this simulation, select the **VELOCITY** and **VOLUMETRIC\_FLOW** datasets. When completed, the Postprocessing Output Panel should appear as in Figure 1.

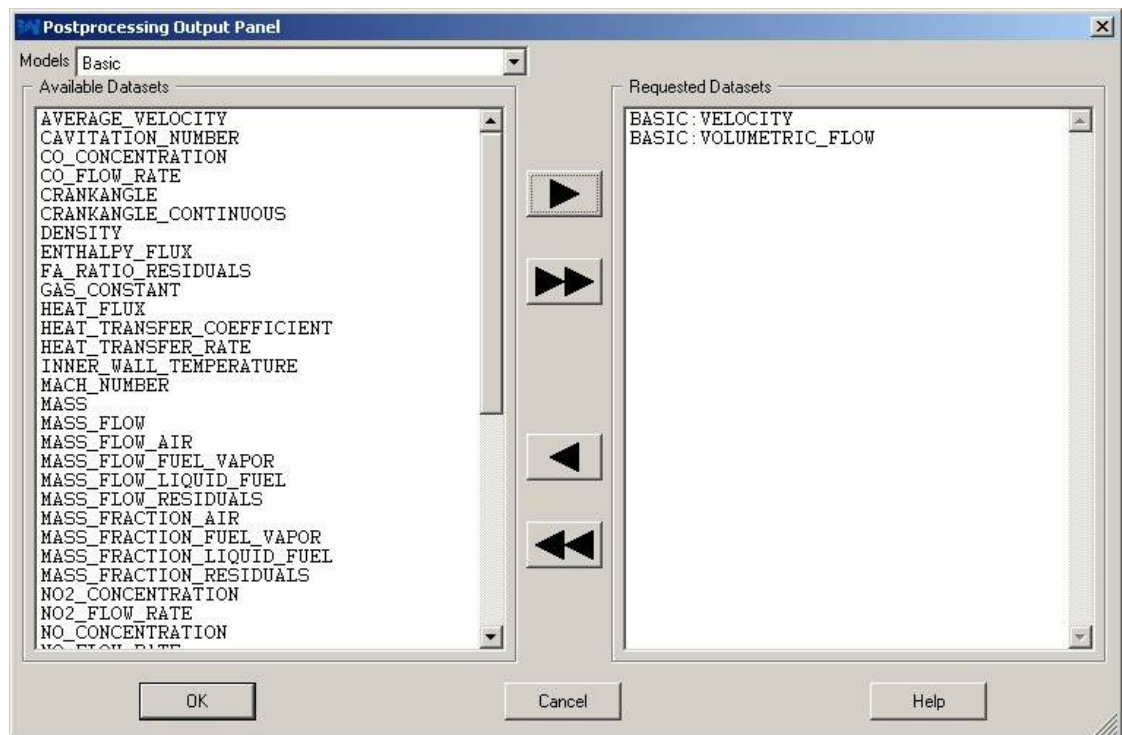


Figure 1: Requesting Basic Datasets

### 3.2 Valve Datasets

At the top of the Postprocessing Output Panel, change the Models option menu to Valve. This will make the Available Datasets list change to those contained in the **Valve** subset. All subsets besides the Basic subset are available depending upon the junctions or physical sub-models included in the analysis.

Highlight and add **DISCHARGE\_COEFFICIENT**, **FLOW\_COEFFICIENT**, and **LIFT\_OVER\_DIAMETER** to the Requested Datasets list. This will allow us to observe the difference between the coefficient used and the one not-selected (CD vs. CF). When completed, the Postprocessing Output Panel should appear as in Figure 2.

Click on the OK button to save the changes and close the panel.

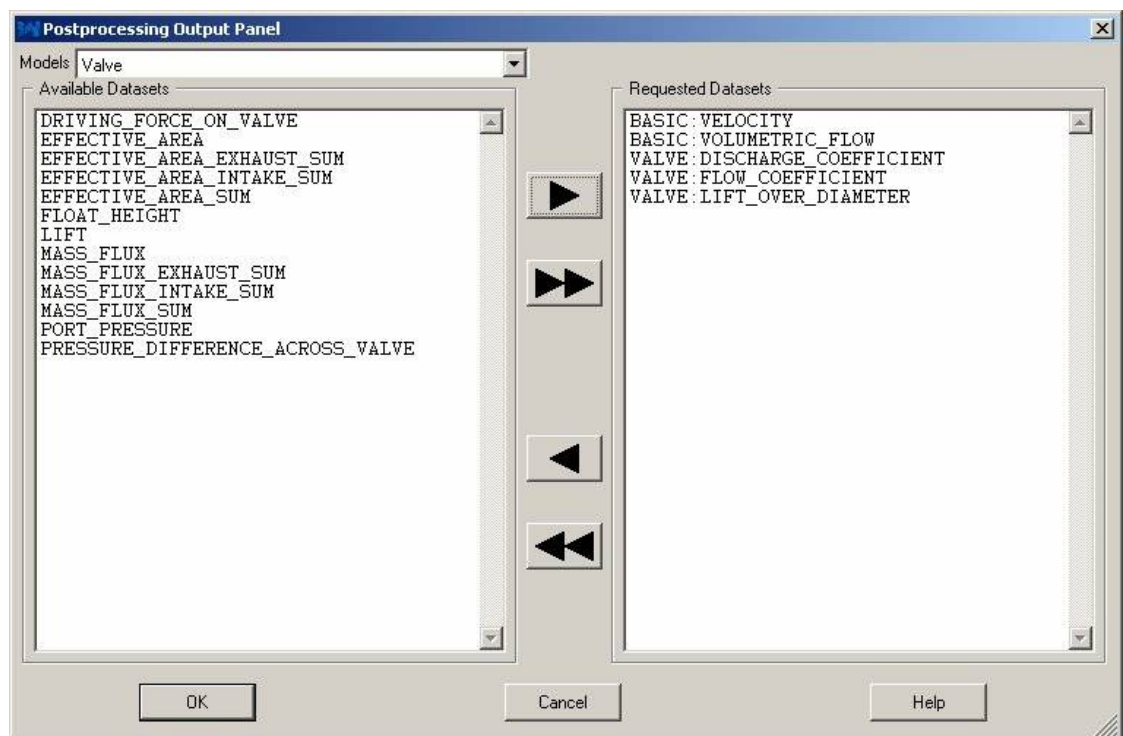


Figure 2: Requesting Valve Datasets


#### Save your model

## 4. Running WAVE and Understanding the .out File

In this step we will run the model file through the WAVE solver and learn to parse the output file. In this and the following step, all output files created during the simulation will be examined and defined as to their purpose and use.

### 4.1 Running the Solver

The model has been run through an input check and is all set to create the requested data for post-processing.

- Run the model in screen mode by clicking on the Run Screen Mode button  in the toolbar

Screen mode runs the model at high priority while sending standard output to the screen. A shell will open and the simulation will pass by, printing output in real-time for the user to examine (output is also printed to the **.out file**).

### 4.2 The Solver

While the simulation is running, WAVE will create temporary files with .mon and .hlt extensions. On all platforms, during the simulation, the progress can be monitored by viewing the contents of the .mon file using a text editor. It is updated every time a cylinder finishes a cycle and the whole file is deleted by WAVE when the run completes. The .hlt file will not be discussed here but both of these files will be automatically deleted upon successful completion of the simulation. If either of these files exists when the simulation completes, it usually indicates that the simulation failed while running in WAVE.

To this point, we have been building the model and saving it along the way into a file with a .wvm extension. It is a simple XML format file that can be observed in any web browser by changing the filename extension to .xml. There should now also exist **.out**, **.sum**, **.wvd**, and **.wps** files as well (a .rp file will have also been created for reverse compatibility with an older post-processor and will not be discussed -- it is obsoleted by the **.wps file**).

## 5. Introduction to Post-Processing and WavePost

In this step we will examine the files created by WAVE for post-processing and use WavePost to examine and create Time Plots from the simulation. We will also observe how WavePost displays Cycle Average quantities using datasets requested in WaveBuild.

Chapter sections Example Input File:


### 5.1 The .sum file

### 5.2 The .wvd file



### 5.3 The .wps file

### 5.4 Time Plots in WavePost.

Launch WavePost from WaveBuild by clicking on the **WavePost** button in the toolbar . The WavePost GUI will open and automatically load the .wps created by WAVE. The network should appear in the main WavePost window identical to its appearance in WaveBuild (see **Figure 1**).

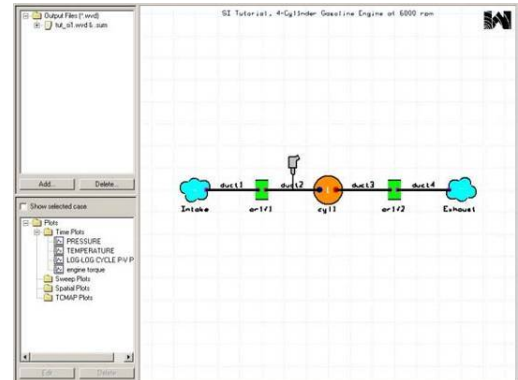


Figure 1: WavePost GUI

The Plot List in the lower right corner of WavePost shows all plots available in the current session file. Plots are categorized as **Time Plots, Sweep Plots, Spatial Plots, or TCMAP Plots**. The Time Plots that we requested in WaveBuild have been automatically created and are listed in the tree under the Time Plots folder. Double-click on the **PRESSURE** plot to see the result (see Figure 2). Notice that this plot has a data line for the single element in **duct2** as well as both of the elements in **duct3**, where we decided to "Use All Locations".

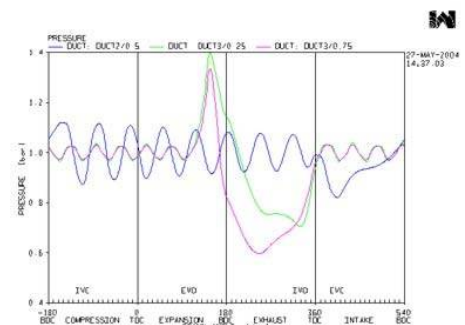



Figure 2: Time Plot of Pressure in duct2 and duct3

Each plot can be individually opened and edited. Elements of the plot, such as the data line, the axes, the title, etc. are all selectable and changeable. Individual data curves can be hidden through the Curve Selector Panel (**Tools -> Curve Selector...**). Data can also be Cut, Copied, and Pasted between plots. A single plot can also be Cloned (**File -> Clone**) to create an exact copy of a plot to use as a template for new data.

Plots can be printed directly to a printer or to an image file by clicking on the Print button  in the toolbar.

Right-click on the **Time Plots** folder and select the **Add Time Plot...** option—a blank time plot window will open. Double-click on the Time Plot #5 title to open the Annotation Panel. Type **"Intake Valve Coefficients"** in the Heading text box and set the **Font** to **Duplex** and the **Size** to **15**. Click **OK** to close the Annotation Panel. Select the **Data...** option from the **Add** pull-down menu to open the Time **Data** Panel. In the Output Sets option menu, highlight the single set that is available (named **filename.wvd:Case 1**). In the Independent Variable (X) section of the panel, select the Custom option and then click on the Edit... button to open the X Axis Selector Panel. Highlight the **Junction Cyl1 Intake 1** option in the Elements option menu and select **Valve Lift Over** Diameter in the Variables option menu. The **X Axis Selector** Panel should appear as in **Figure 3**, right. Click the OK button to save the selections and close the panel. Back in the Time Data Panel, under the Elements option menu, highlight the **Junction Cyl1 Intake 1** option and pick **Valve Flow Coefficient** in the Variables option menu. When finished, the Time Data Panel should appear as in **Figure 4**.

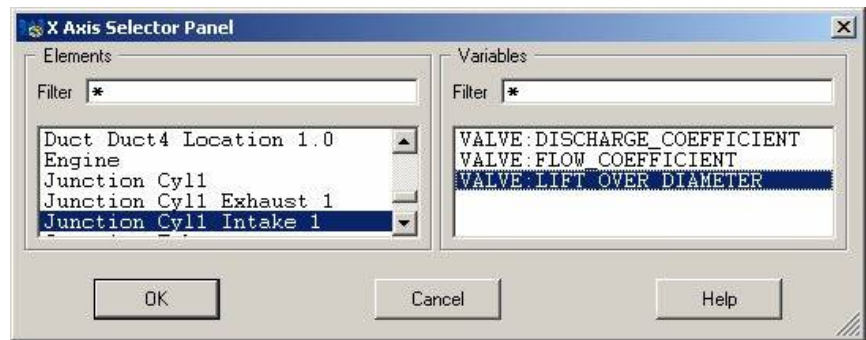


Figure 3: X Axis Selector Panel

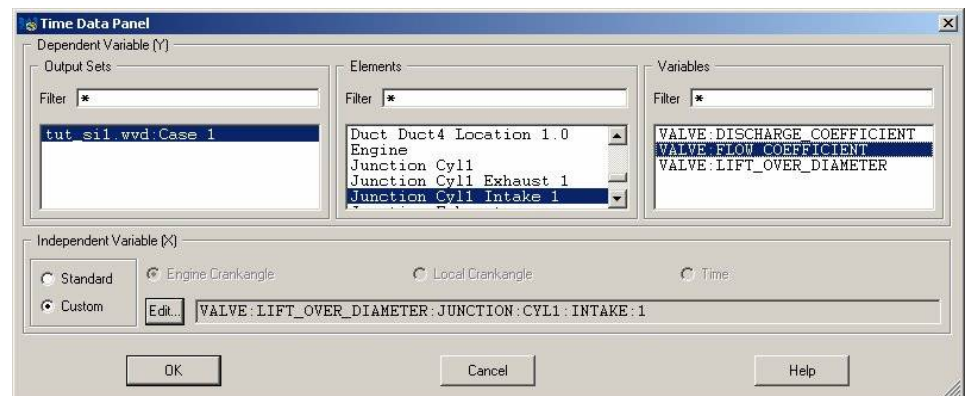


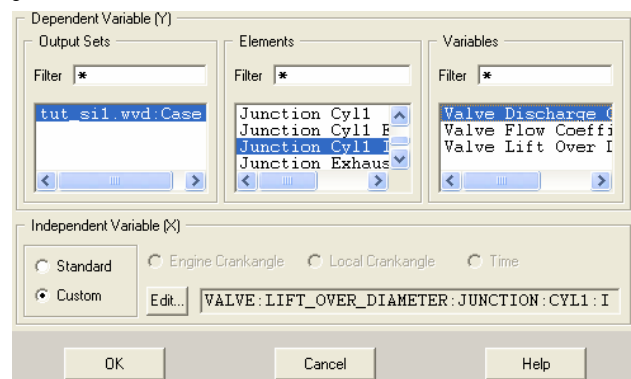


Figure 4: Time Data Panel

Click the OK button to save the settings and close the panel. The data curve for Flow Coefficient will appear in the Time Plot and the plot title and axis labels will be automatically generated. Double-click on the plot **title** and edit it to read "**Cf vs. Cd**" and set the Font to Duplex. Double-click on the Y-axis and edit the label to read "Valve Coefficient" (delete the word Flow) and set the Font to Duplex. Double-click on the X-axis and edit the label to read "L/D" and set the Font to Duplex. Double-click on the plot frame (easiest to do at the top or right-edge of the plot) and click in the Grid checkbox.

Highlight the data curve and, using the Copy  and Paste  toolbar buttons, paste a second data curve on the same plot. Double-click on the second data curve in the legend to open the Curve Panel. Click on the Edit Data button and then click on the Modify Data Source button. Change the Variable to **VALVE:DISCHARGE\_COEFFICIENT** and click on the **OK** button to save the change.



Don't forget to update the Legend in the Curve Panel to reflect the change from flow to discharge coefficient. When finished, the Time Plot should appear as in Figure 5. Close the Time Plot and note that a fifth plot is in the list under the Time Plots folder named "Cf vs. Cd".

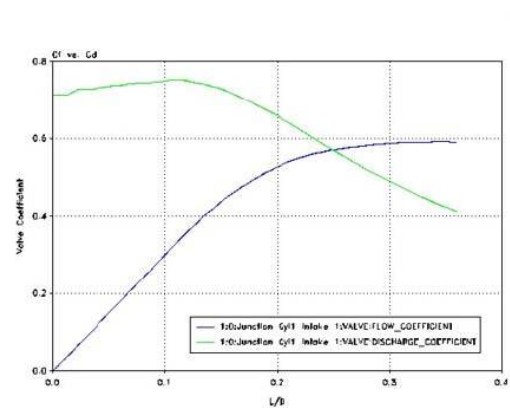


Figure 5: Cf vs. Cd Plot for cyl1 Intake Valve

Time plots can also be made quickly by **right-clicking** on an **element** in the flow network diagram and selecting a **variable to plot**. Right-click on **duct1** to create a time plot of **Velocity** at location **0.0** (Figure 6) and right-click on **cyl1** to create a time plot of **Pressure** (Figure 7).

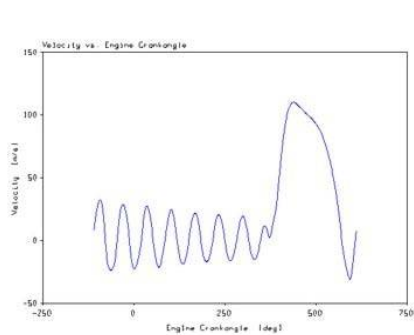


Figure 6: Velocity Time Plot at duct1, Location 0.0

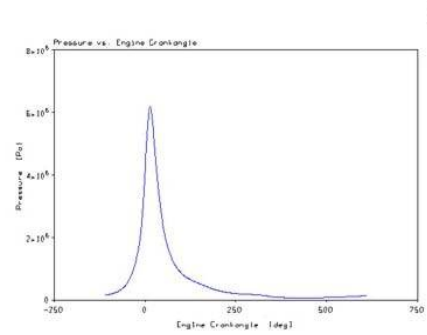


Figure 7: Cylinder Pressure Time Plot at cyl1

### 5.5 Cycle Average Results in WavePost

Any dataset that stores data for elements (cells/junctions) which was requested in WaveBuild can be displayed as color contours on the flow network diagram. Select the **Cycle Average** option from the **Mode** pull-down menu. Open the **Variable Panel** by selecting **Variable...** from the Tools pull-down menu. In the Variable Panel, select Velocity and observe how WavePost changes the flow network diagram. The ducts and junctions are redrawn with relative diameters displayed and color contours to represent the selected variable.

The scale at the bottom of the window automatically sets upper and lower bounds by using the highest and lowest calculated values from the dataset. These can be controlled in the Variable Panel using the slider bars or by typing numerical values in the given text fields.

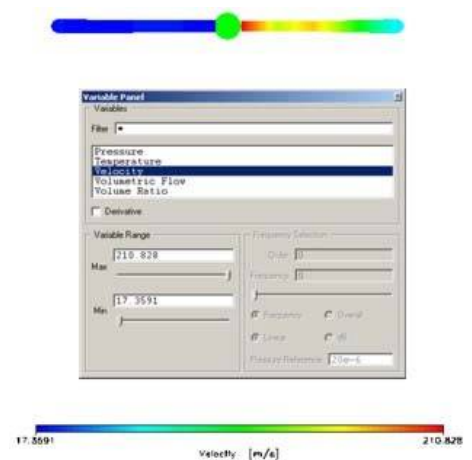
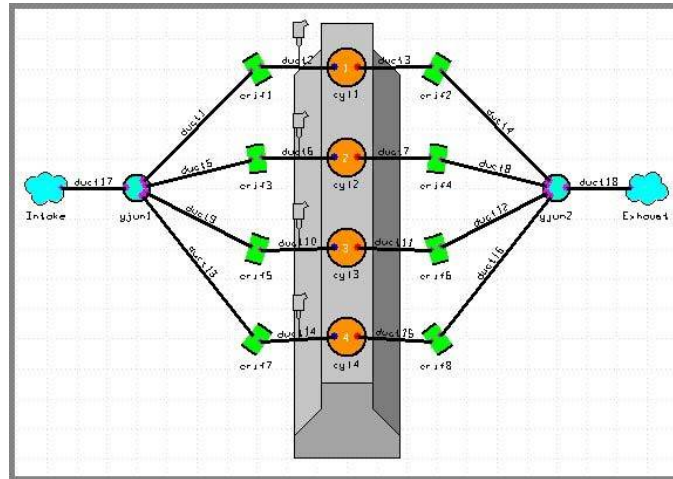


Figure 8: Cycle Average Velocity

# SI Tutorial, Phase 3 - Multi-Case, 4-Cylinder Model and Sweep Plots in WavePost



## 1. Copying and Pasting the Single-Cylinder Model

Time and effort can be saved when creating our 4-cylinder model by using the existing single-cylinder model as a template. Copy and Paste functionality can be used to **create three copies of the single-cylinder** network, including the attached ducts representing the ports. The engine information then needs to be updated to reflect the addition of three new cylinders.

### 1.1 Detaching the Ambient Junctions

Using the middle mouse button, drag the Intake ambient further to the left, to move it out of the way. Repeat for the Exhaust ambient, moving it further to the right. When finished, the model should appear as in **Figure 1**.

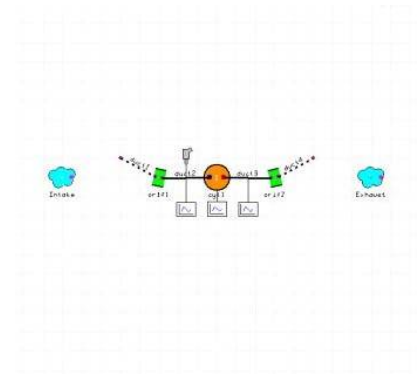


Figure 1: Ambients Detached from Ducts

### 1.2 Copying and Pasting the Network

Using the left mouse button, draw a box around the ducts and engine-cylinder network to select the entire system (be careful not to draw the box around the ambient junctions). All of the selected items within the box will be highlighted in red. Click on the Copy button in the toolbar . Then click on the Paste button in the toolbar and the mouse pointer will become a crosshair icon. Click on the canvas beneath the cy11 junction and a duplicate network will be created. Click on the Paste button and place the duplicate network two more times to create four identical duct/junction networks to represent all four engine-cylinders. Note that the ducts and junctions have all been numbered sequentially.



Figure 2: Canvas Properties panel

Plot requests are not duplicated, thus no plots are dangling off of any of the newly created ducts/cylinders (feel free to request new plots if desired). To hide the plots that currently exist on the network, right-click anywhere in the white canvas area and select the Edit Canvas Properties... option from the menu to open the Canvas Properties panel. De-select the **Plots** toggle button in the Annotations section of the panel (see Figure 2). This will simply hide the plot icons and not draw them on the canvas. When finished, the model should appear as in Figure 3.

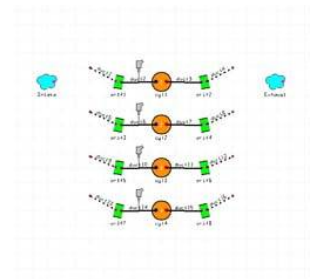


Figure 3: Duct/Junction Network Duplicated

### 1.3 Creating an Engine Block Icon

Select the **Create Engine...** option from the **Tools** pull-down menu to open the Create Engine Panel. This panel displays currently entered geometric values from the **Engine General Panel**. Currently, there is only one cylinder in the engine -- this must be updated to reflect that three new cylinders have been added. Change the **No. of Cylinders** text field to **4** and press the **Enter** key. This will update the Preview of the engine block on the right as well as the Firing Order table at the bottom. The Firing Order table will automatically calculate the TDC (top dead center time) for each cylinder based upon the **No. of Cylinders** value and the **Strokes per Cycle** selection (TDCs are calculated for even firing intervals and are relative to the previous cylinder, with the first firing cylinder at crank-angle 0). Change the **Firing Order** to reflect that of a standard 4-cylinder engine -- **1, 3, 4, 2**. The default spacing of the cylinder TDCs will be appropriate for this tutorial. When completed, the Create Engine Panel should appear as in Figure 4. Click the OK button to close the panel and note the Engine Block icon that is added to the canvas.

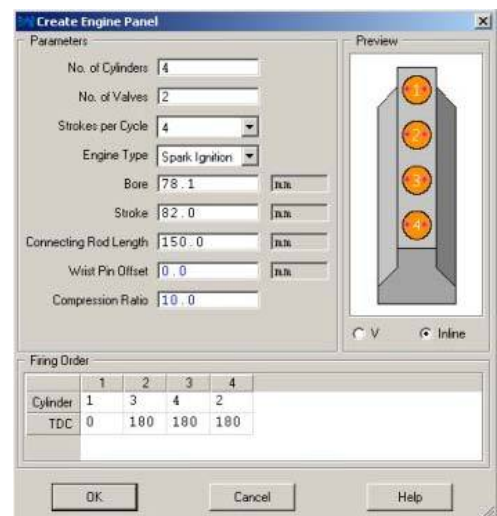


Figure 4: Create Engine Panel

When the **Engine Block** is created, it will have four **Engine Cylinder** junctions created along with it by default. **Left-click** each of these **newly-created Engine Cylinders** one at a time (they will highlight in red) and press the **Del** key to delete them, leaving an empty **Engine Block** icon. Move the **Engine Block** icon over to the **Engine Cylinders** that are currently on the canvas **by middle-clicking** on the icon and **dragging** it. The existing **Engine Cylinder** junctions can be "dropped" into the icon by middle-clicking on them, one at a time, and placing them over the cylinder place-holders on the icon.

They will snap into place on the icon and be associated with the icon from that point on. The **bore-spacing** of the **Engine Block** icon can be adjusted by **right-clicking** on the icon and selecting **Appearance...** from the menu. Default WaveBuild grid spacing is 40/square so, if the cylinder junctions are placed 3 grid squares apart, use a spacing of **120** (see Figure 5).

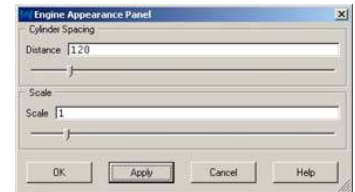


Figure 5: Engine Block Icon Appearance

When completed, the model should appear as in Figure 6.

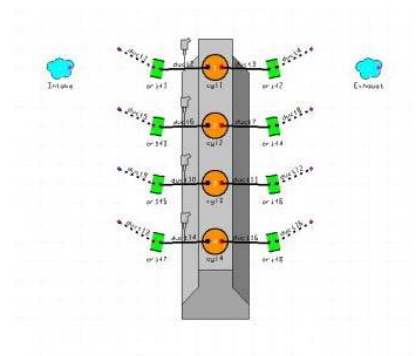



Figure 6: Model with Engine Block Icon

 **Save your model**

## 2. Joining the Cylinders Using a Simple Y-Junction

A simple Y-junction will be used to join all four engine-cylinders together at the intake ports. This will represent a large spherical plenum with a single intake pipe that will connect to the existing Intake ambient junction. The same will be modeled on the exhaust side.

### 2.1 Placing the Simple Y-Junctions on the Canvas

Open the **Junction Palette...** under the **Tools** pull-down menu. Left-click on the Simple Y-junction button  and place one Y-junction on each side of the engine near the middle, vertically. Press the **Esc** key to exit junction placement mode and return to select mode (**see Figure 1**).

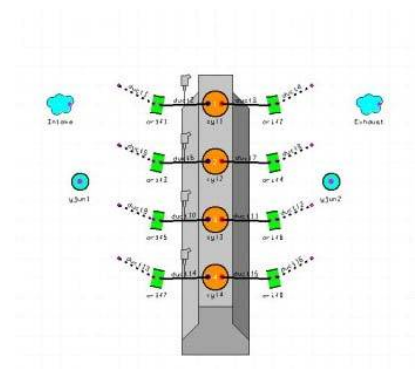



Figure 1: Simple Y-Junctions on the Canvas

**Connect the dangling duct ends to the Simple Y-junction** by dragging and dropping (use the middle-mouse button) anywhere on the blue portion of the junction. Any duct end dropped onto the blue portion of the junction will create its own connection point automatically. Dropping the dangling duct end on the existing connection point,  , will occupy that connection point, leaving no starting point to draw a duct away from the Y-junction. If there are no connection points on a Y-junction and a duct must start at that junction and be drawn away from it (to follow the **Left to Right convention**), simply left-click on the blue portion and drag a duct away from the Y-junction.

**Create a new duct** between the **Intake** ambient and the **Simple Y-junction** on the intake side. Enter **50 (mm)** for both **Left and Right Diameters** and **500 (mm)** for **Overall Length**. The **Discretization Length** should be **35 (mm)**, as used earlier in the single-cylinder model. The default initial conditions are suitable for this duct.

**Create another new duct between the Simple Y-junction on the exhaust side and the Exhaust** ambient (following the **Left to Right convention**). Enter **50 (mm)** for both Left and Right **Diameters** and **500 (mm)** for **Overall Length**. Enter **40 (mm)** for the **Discretization Length**. Appropriate initial conditions for this duct should be set as **1.05 (bar)** Pressure, **700 (K)** Temperature, and **650 (K)** Wall Temperature.

When completed, the model should appear as in Figure 2.

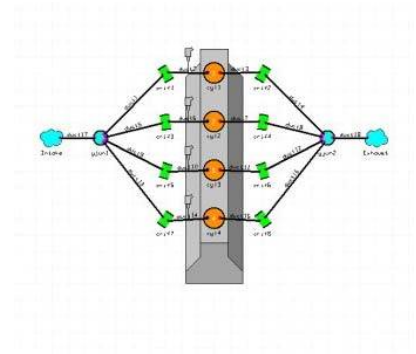


Figure 2: All Ducts Connected

## 2.2 Defining the Simple Y-Junctions

The Simple Y-junctions must be fully defined before the model will run. We must define the geometry of the junction as well as the orientation for all of the connected ducts.

**Double-click** on the intake-side Y-junction to open **the Simple Y-Junction Panel**. The geometry for the Y-junction is defined using only a Diameter value, as the junction is assumed to be spherical in shape. The volume and surface area are therefore easily determinable using the entered Diameter value. Type **50 (mm)** in the **Diameter** text-field. The default coefficients and initial conditions are suitable for this tutorial (see Figure 3).

Figure 3: Intake-side Simple Y-Junction Panel

**Click** on the Edit Openings button to orient the connected ducts.



The junction/ducts can be rotated in 3-D space by holding the Shift button while clicking and dragging in the window using the middle mouse button.



For each attached duct, the orientation needs to be given using three angles to describe the duct position relative to the X, Y, and Z axis.

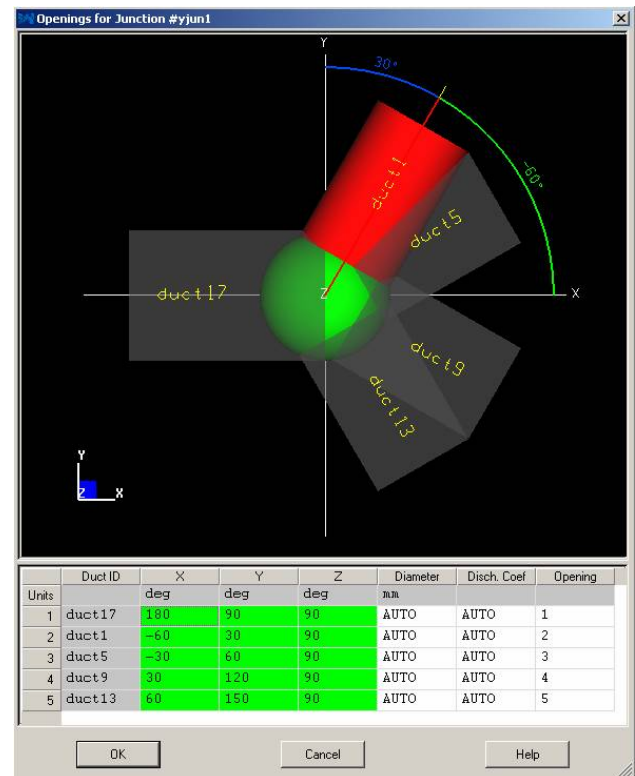


Figure 4: Openings Panel for Simple Y-Junction 1

**The exhaust-side Simple Y-junction** should be set up similarly, with a Diameter of **50** (mm) and initial conditions similar to the outlet duct -- Pressure of **1.05** (bar), Temperature of **700** (K), and Wall Temperature of **650** (K). **The orientation** of the ducts should be **similar** to that of the layout on the canvas, as in the intake-side Y-junction. When completed, the Simple Y-Junction Panel and Openings panel should appear as in Figure 5 and Figure 6 below, respectively.

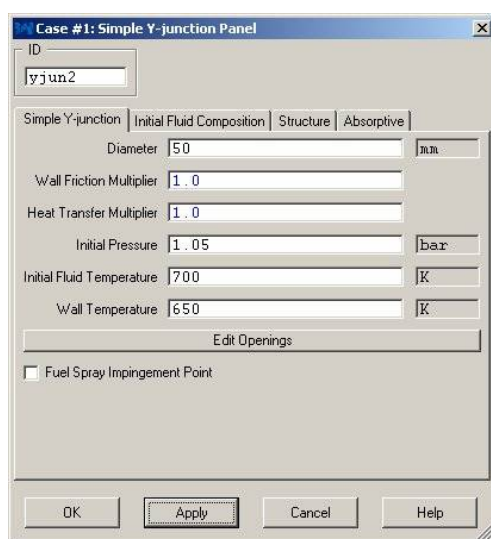
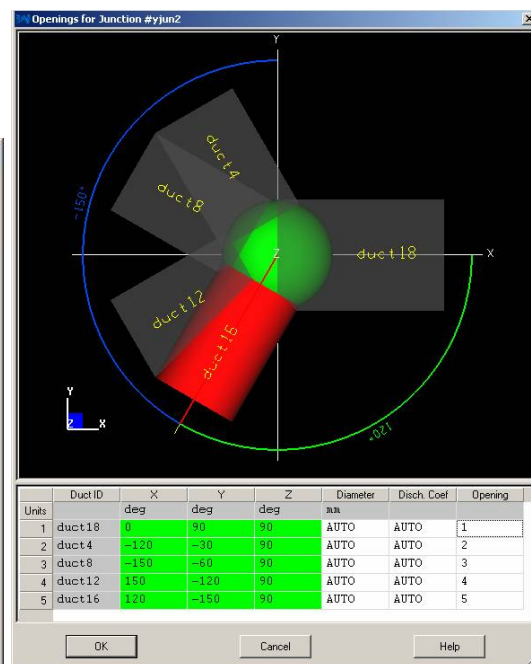


Figure 5: Exhaust-side Simple Y-Junction Panel Figure



6: Openings Panel for Simple Y-Junction 2

**Save your model**



### 3. Creating a Multi-Case Speed Sweep

The model is complete. To get useful results from this model, you need to set up the run conditions

#### 3.1 Adding Cases with the Case Manager

Cases are added and deleted using the Case Manager at the bottom of the WaveBuild canvas. Click on the Add Case button

 5 times to create 5 more cases for a total of 6 cases.

It is dangerous to edit the model in any case other than Case #1 as changing the elements, components, or general behavior between cases can cause WAVE to crash at run-time. Therefore when in any case other than Case #1, the background of the case text field will be red to warn the user as in **Figure 1**.




Figure 1: Case Manager when case other than Case #1 is selected

Make sure to return to Case #1 before continuing by either typing directly into the text field or using the arrow selection buttons.

#### 3.2 Changing Constants between Cases

The convenience of using multiple cases is that Constant values can be changed from case to case.

For this tutorial, we will step from 6000 rpm down to 1000 rpm, using 1000 rpm increments to simulate multiple steady-state test points in a speed sweep. Open the

Constants Panel  and note that there are now columns added to the table for every new case created.

Select the **Case #2** field for the SPEED constant and enter 5000. Enter **4000 - 1000** for **Cases #3 - #6**, respectively. The Constants Panel should appear as in **Figure 2**. Click OK to close the panel and save the settings.

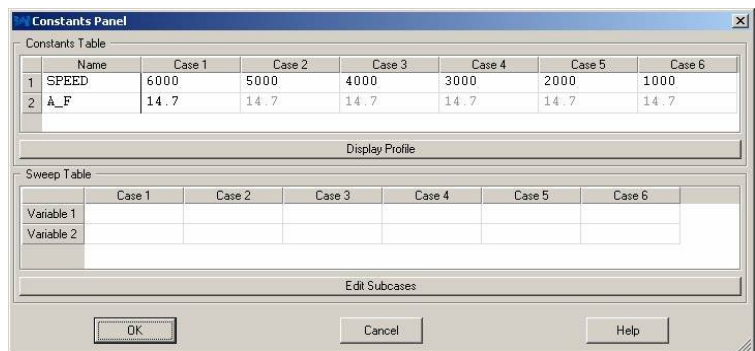


Figure 2: SPEED values in Constants Panel

When running a speed-sweep simulation, it is recommended to start at the high speed and move downwards toward the low speed. This is because the WAVE solver is actually working in a timebase of seconds and the solution tends to converge based on the number of repeated engine cycles. More cycles can be completed at high RPM in a given amount of time in seconds than at low RPM. This means a system running a high RPM will tend to finish quicker than at low RPM, thus any problems with the general setup may be detected earlier when starting at a high RPM.

Typically, with a change in engine speed other parameters change as well, such as combustion behavior and cylinder temperatures.



Figure 3: Undefined Constants Message Box

**Double-click** on the **Engine Block** icon to open the **Engine General Panel**. Click on the **Heat Transfer** tab and enter **{PISTON\_TEMP}**, **{HEAD\_TEMP}**, and **{LINER\_TEMP}** in the text fields for the Piston Top, Cylinder Head, and Cylinder Liner temperatures, respectively. Click on the Apply button and a message will appear

as show in **Figure 3**.

WaveBuild has detected that new constants have been used but are not defined in the Constants Panel. Click on the Yes button to open the Edit Constants panel and edit the profiles for these three new constants. Enter the values as shown in **Figure 4**. These profiles describe the temperature of the combustion chamber cooling slightly with a decrease in engine speed. Click OK to save these constant profiles and close the Edit Constants panel.

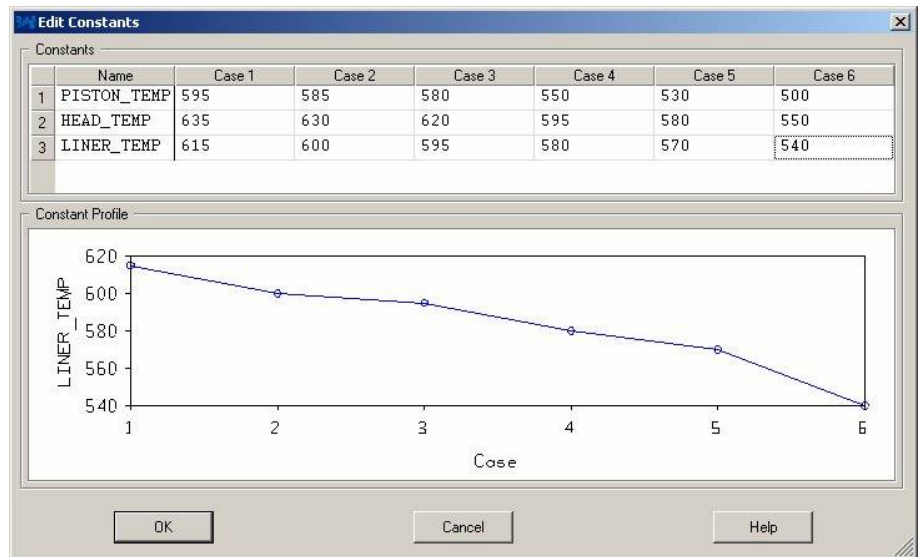


Figure 4: Profiles for Combustion Chamber Temperatures

Click on the **Combustion** tab and enter **{CA50}** in the **Location of 50% Burn Point** text field and **{BDUR}** in the **Combustion Duration (10-90%)** text field. Click on the **Apply** button again to be queried on adding these constants to the table. Select Yes from the Query window and enter the profiles as given in **Figure 5**. These constants help to describe the shorter crank angle duration of combustion and retarding of spark timing at lower engine speeds. Click **OK** to save these constant profiles and close the Edit Constant panel. Click the **OK** button to close the Engine General Panel.

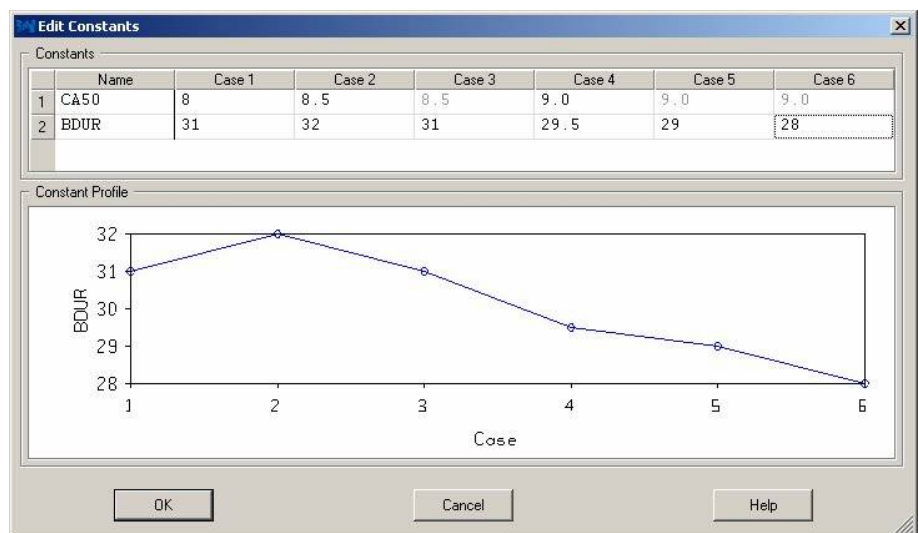



Figure 5: Profiles for SI Wiebe Function Inputs

**Save your model**

Click on the Save button in the toolbar to save the file and then click on the Run Screen Mode button to launch the analysis (note how engine cycles take longer to complete in the later cases, at lower engine speeds).

## 5. Creating Sweep Plots in WavePost

### 5.1 Multi-case Handling in WavePost

Launch WavePost from WaveBuild by left-clicking in the Launch WavePost button in the WaveBuild toolbar .

Selecting a case in this way will also change the behavior of displayed results. **Manually-created Time Plots** and cycle-averaged results or animations will display results from the selected case.

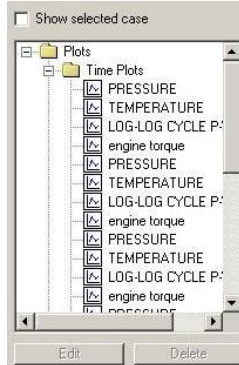


Figure 1: WavePost Plot Tree

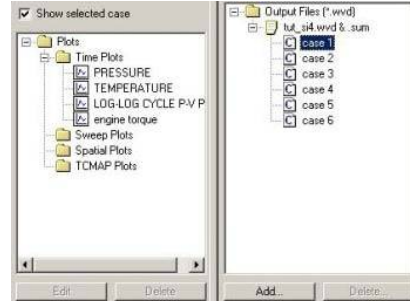


Figure 2: Filtering Plots by Case

### 5.2 Creating Sweep Plots

To create a sweep plot, **right-click** on the **Sweep Plots** folder in the Plots Frame and select the **Add Sweep Plot -> 2D Plot...** menu items (see **Figure 3**). This will create a new Sweep Plot named Sweep Plot #1 by default. The plot is empty, with no data displayed. To add data, select the **Add pull-down menu** and then select the **Data...** menu item to open the Sweep Data Panel. This contains a list of all of the keywords in the .sum file that correspond with a cycle-averaged numeric value. The [list defining sum file keywords and their associated units](#) is available in the REFERENCE LIBRARY, under the Tables section. Select bhp for the Dependant Variable (Y) and select speed for the Independent Variable (X). The Sweep Data Panel should appear as in **Figure 4** when completed. Click the **OK** button to apply.

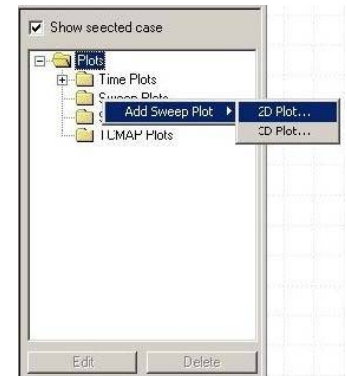


Figure 3: Creating a New Sweep Plot

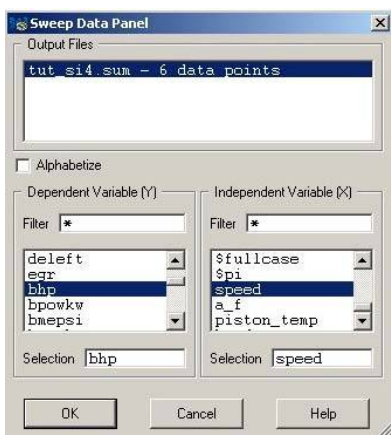



Figure 4: Sweep Data Panel

 Save your file

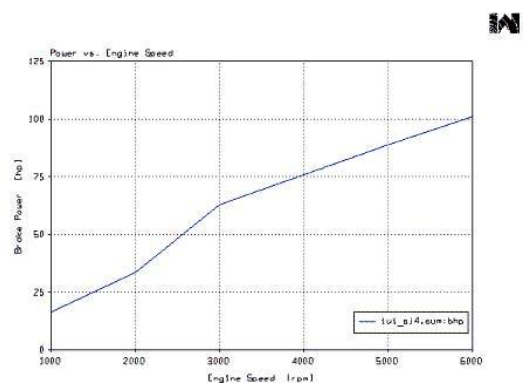
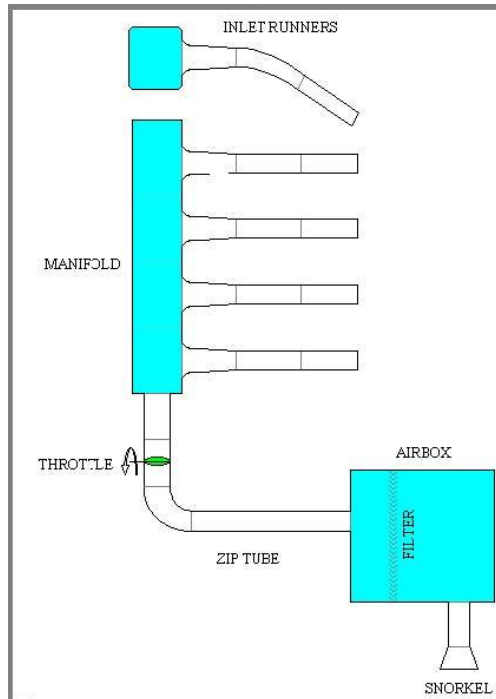


Figure 5: Power vs. Engine Speed

## SI Tutorial, Phase 4 - The Intake System and Animations in WavePost

The Intake or Induction system consists of a snorkel, air cleaner with filter, zip tube, throttle body, and intake manifold. All of these parts will be modeled in WaveBuild to create the induction system with appropriate restrictions.



### 1. The Snorkel and Zip Tube

#### 1.1 Detaching the Intake Ambient Junction

Highlight both the intake-side **Simple Y-junction** and upstream duct (holding down the shift key and left-clicking) and hit the **Delete** key. This will leave all four intake port ducts dangling with no junction to attach to upstream (**see Figure 1**). These will eventually be attached to the ends of the runners from the intake manifold. For now, they can remain unattached.

Click and hold the middle mouse button to move the ambient junction named Intake down to the bottom of the canvas. This will serve as the starting point for the induction system.

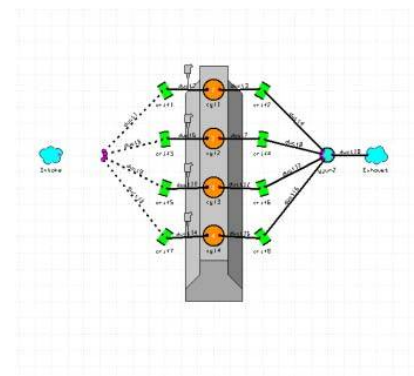



Figure 1: Y-Junction and Upstream Duct Deleted

#### 1.2 Placing the Required Orifice Junctions and Ducts

The snorkel and zip tube can easily be modeled using only orifice junctions and ducts. A schematic of the system with required dimensions is shown in Figure 2. Place

orifice junctions and connect with ducts as shown in Figure 3. In place of the air cleaner, place two Complex **Y-junctions**  and connect with a duct. Remember to follow the Left to Right convention when creating the ducts.

To make the last duct in the zip tube (**duct22 in Figure 3**) appear bent on the screen, right-click on the duct and select **Add Control Point** from the menu:

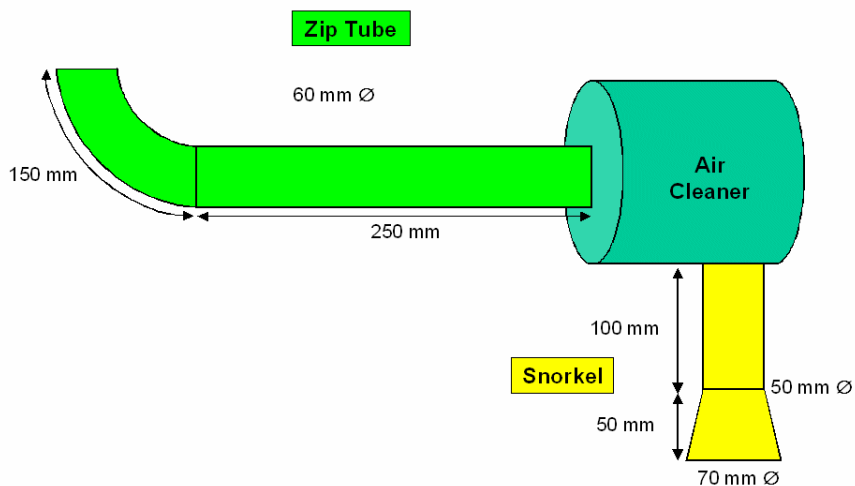


Figure 2: Snorkel and Zip Tube Schematic

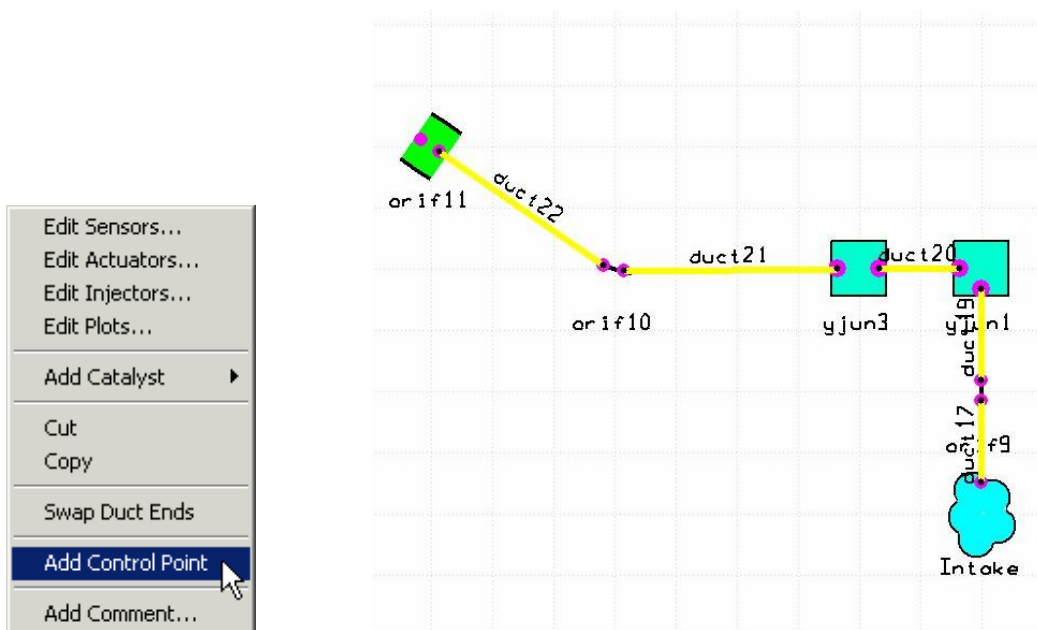


Figure 3: Duct/Junction Network Duplicated

A control point can be selected with the **middle mouse** button and moved to make the duct appear bent (it will snap to grid points just like junctions do). Multiple control points can be added to a duct. Using two control points diagonal from each other by one grid square will create the appearance of a **smooth 90° bend**, as shown in Figure 4.

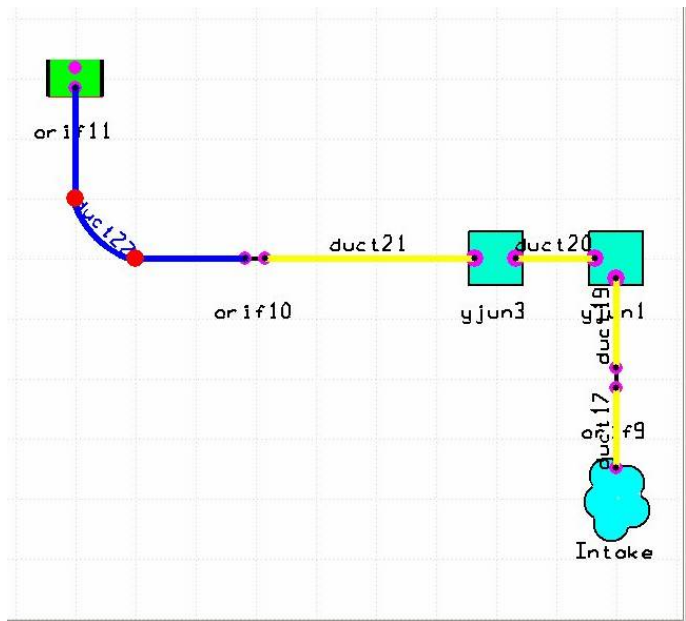


Figure 4: Control Points to Draw Bent Duct

### 1.3 Defining the Ducts (Tapered and Bent)

Using the schematic from **Figure 2**, above, edit the **ducts** representing the snorkel and zip tube and enter the appropriate **Diameters** and **Lengths** (leave the Complex Y-junctions and duct between them for later). Remember to use **35 (mm)** for the **Discretization Length** for all ducts in the intake system and set initial conditions to be **1 bar Pressure**, and **300 K** for Initial **Fluid** and **Wall Temperatures**.

Note that when setting the Left Diameter of the first duct to **70 (mm)** and the Right Diameter to **50 (mm)**, with Overall Length of **50 (mm)**, the Taper Angle field turns yellow, indicating a warning that the calculated taper angle of  $11.3099^\circ$  is outside of the recommended range for this parameter (see **Figure 5**). Read this sidebar on modeling tapered ducts and note that we are modeling upstream of the intake manifold, thus expect constant flow into the engine. Flow will always be contracting and therefore the slightly high taper angle is not of concern.

Also note that when entering dimensions for the last duct in the zip tube, the **90° Bend Angle** should be included (see **Figure 6**, lower left). The Bend Angle field on the Dimensions tab of the Duct Panel allows the user to specify how much of a bend occurs *across the entire length of the duct*. The pressure drop due to this bend is also then distributed across the entire length of the duct.

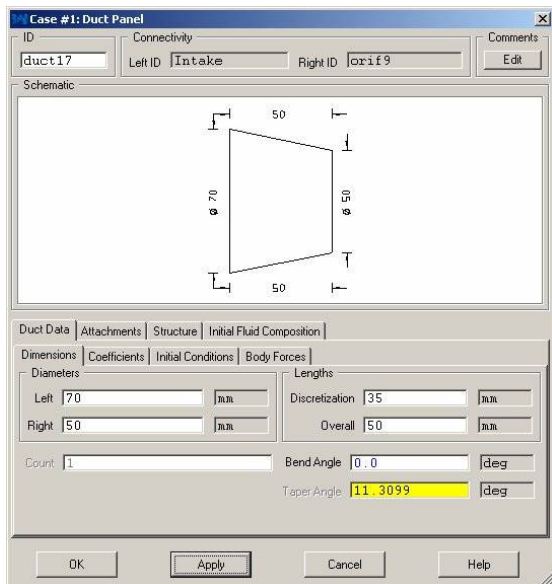


Figure 5: High Taper Angle

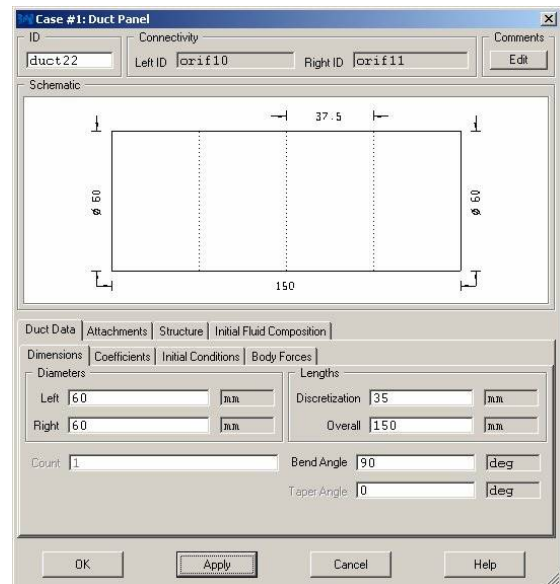


Figure 6: Duct with Bend Angle Specified

Template Name  
Default Duct

ID  
duct21

Connectivity  
Left ID yjun3 Right ID orif10

Schematic

Duct Data | Attachments | Structure | Initial Fluid Composition

Dimensions | Coefficients | Initial Conditions | Body Forces

Diameters  
Left 60 mm Right 60 mm

Lengths  
Discretization 35 mm Overall 250 mm

Count 1 Bend Angle 0.0 deg Taper Angle 0 deg

ID  
duct19

Connectivity  
Left ID orif9 Right ID yjun1

Schematic

Duct Data | Attachments | Structure | Initial Fluid Composition

Dimensions | Coefficients | Initial Conditions | Body Forces

Diameters  
Left 50 mm Right 50 mm

Lengths  
Discretization 35 mm Overall 100.0 mm

Count 1 Bend Angle 0.0 deg Taper Angle 0 deg

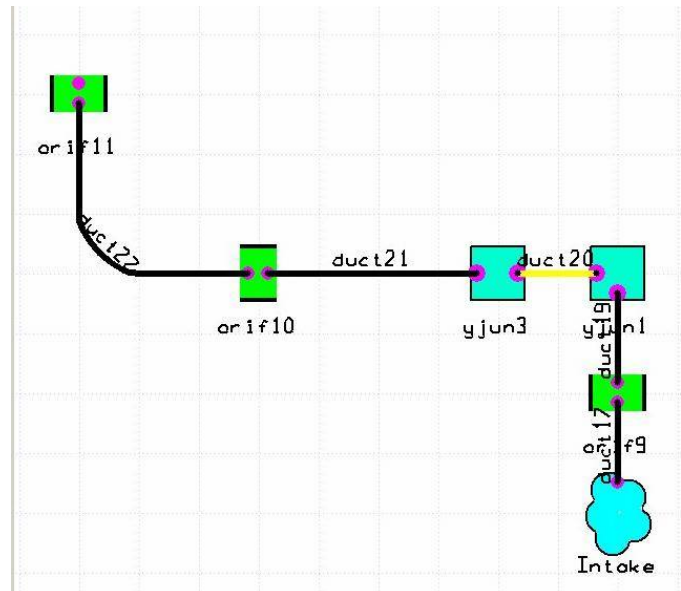


Figure 7: Intake Subsystem

When completed, the intake subsystem should appear as in Figure 7.

Save your model

## 2. The Air Cleaner with Filter

In the last step, two Complex Y-junctions were placed on the canvas with a duct connecting them to represent the air cleaner and filter. In this step, we will define these junctions and duct to appropriately represent the physics of the system in WAVE.

### 2.1 Defining the Complex Y-Junctions

A schematic of the air **cleaner geometry** is shown in **Figure 1**. It is simply two cylindrical volumes separated by a flat, circular filter. It is impossible to appropriately model the filter passages true to life as flow through a fibrous filter is very non-one-dimensional. Typically, a filter is modeled as an orifice that approximates the pressure drop of the real system.

For our model, the two Complex Y-junctions will represent the two cylindrical volumes and the duct connecting them will represent the filter. **Double-click on yjun1** and enter the appropriate values as given in **Figure 2**.

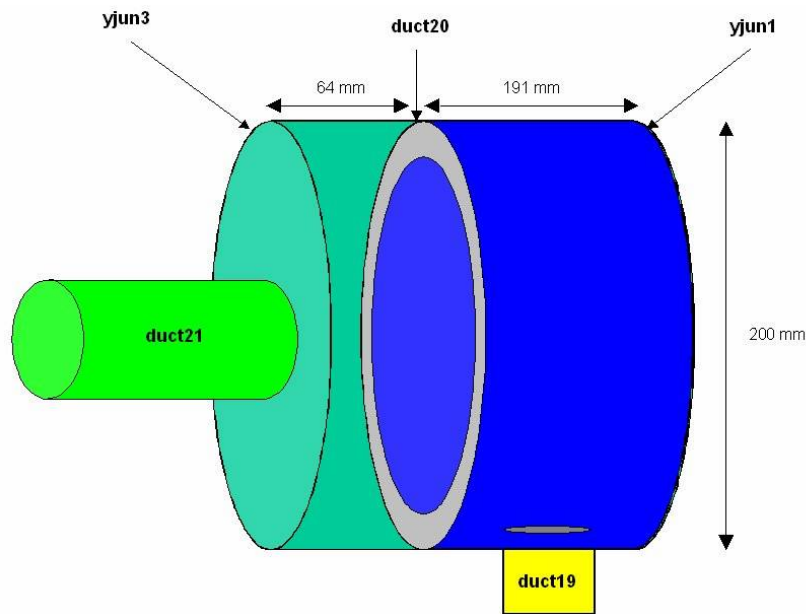


Figure 1: Airbox Schematic

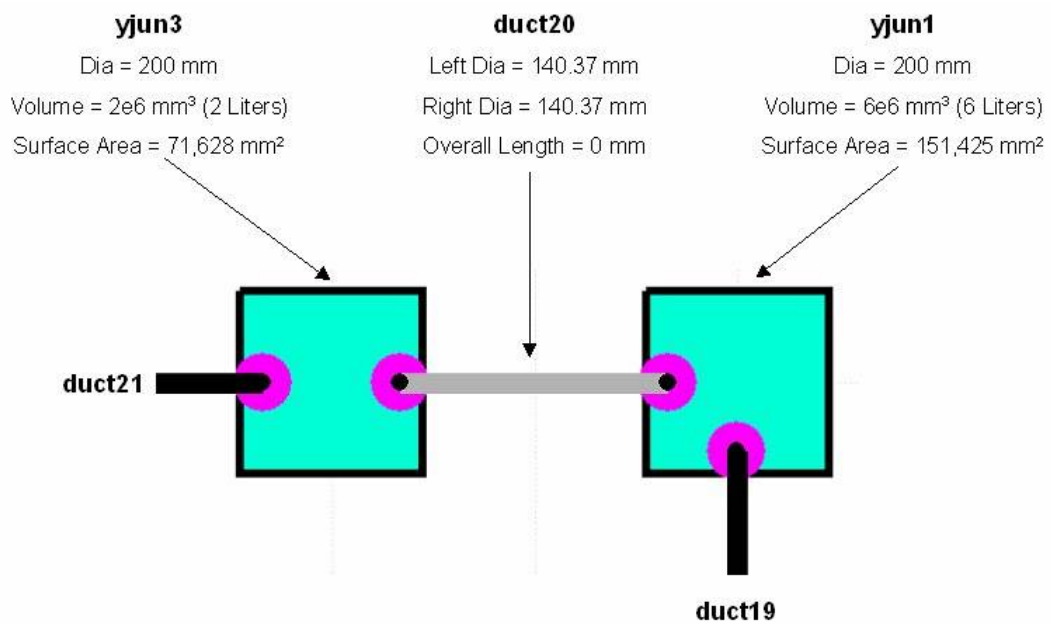


Figure 2: Airbox Network Representation

Click on the Edit Openings button to orient the connected ducts.



Ducts are oriented similarly as on the Simple Y-junction. The orientation of duct19 and duct20 should be used as shown in Figure 1. Note the new fields for each connected duct: DELX, DIAB, and Thick.

DELX, sometimes referred to as the characteristic length, is the distance from the duct connection point across the volume. See Figure 3 for a diagram of the DELX values for both Complex Y-junctions.

DIAB, sometimes referred to as the expansion diameter, is the equivalent diameter for the maximum area that the gas can expand into, perpendicular to the duct entrance. See Figures 4 and 5 for diagrams for the DIAB values for both Complex Y-junctions.

Thick is the orifice thickness and is used in acoustics simulations to calculate the acoustic end correction. It is not necessary to set this value in performance simulations as it has no effect whatsoever on the outcome.

When completed, the orientations for the Complex Y-junctions should appear as shown in Figures 6 and 7.



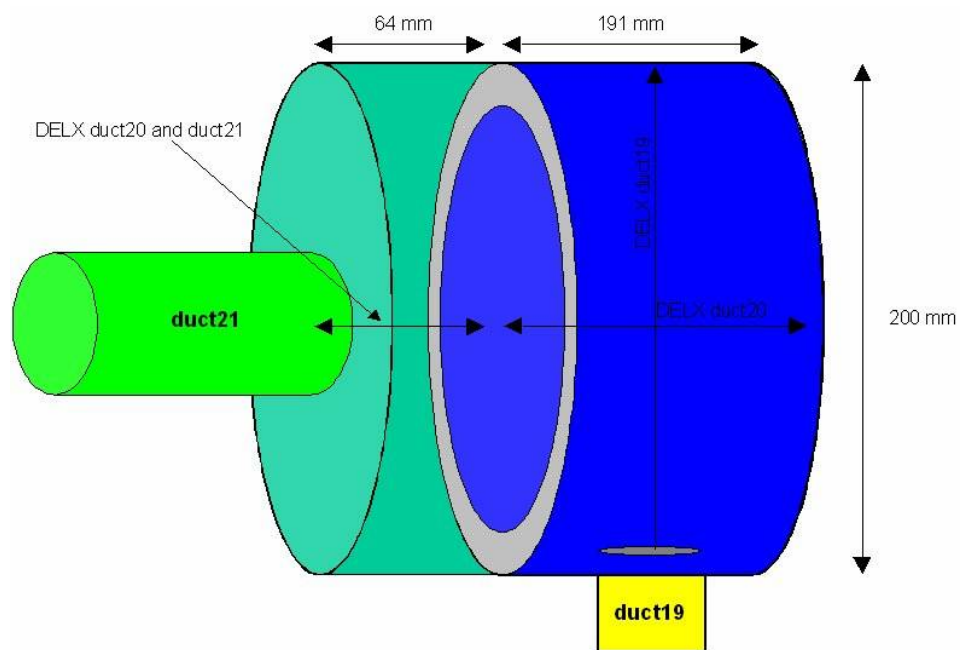


Figure 3: DELX Values

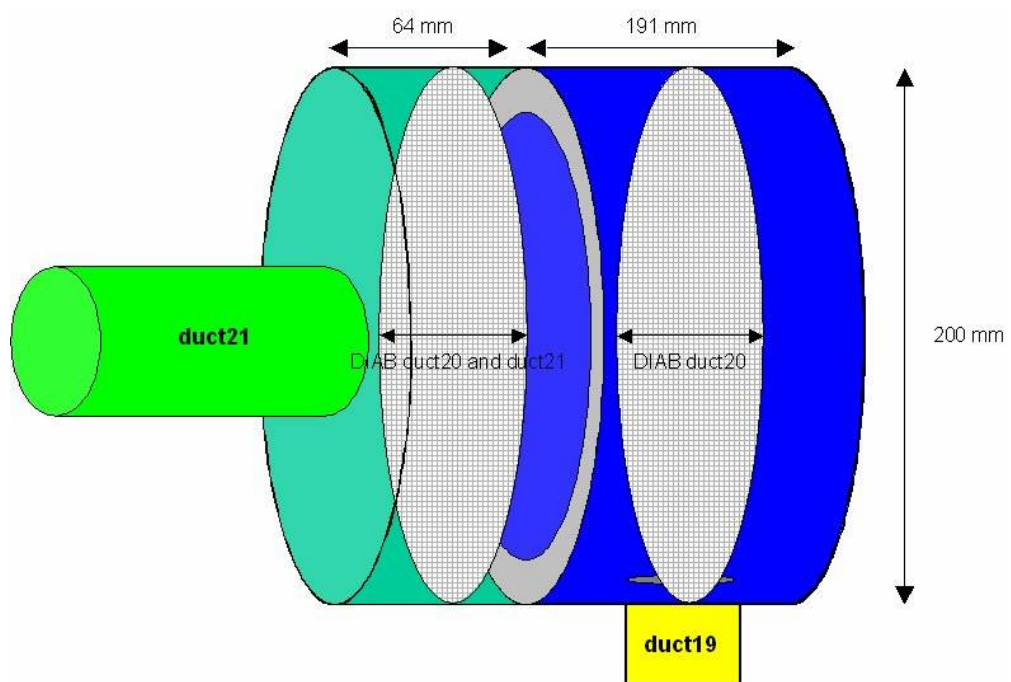


Figure 4: DIAB Values for duct20 and duct21

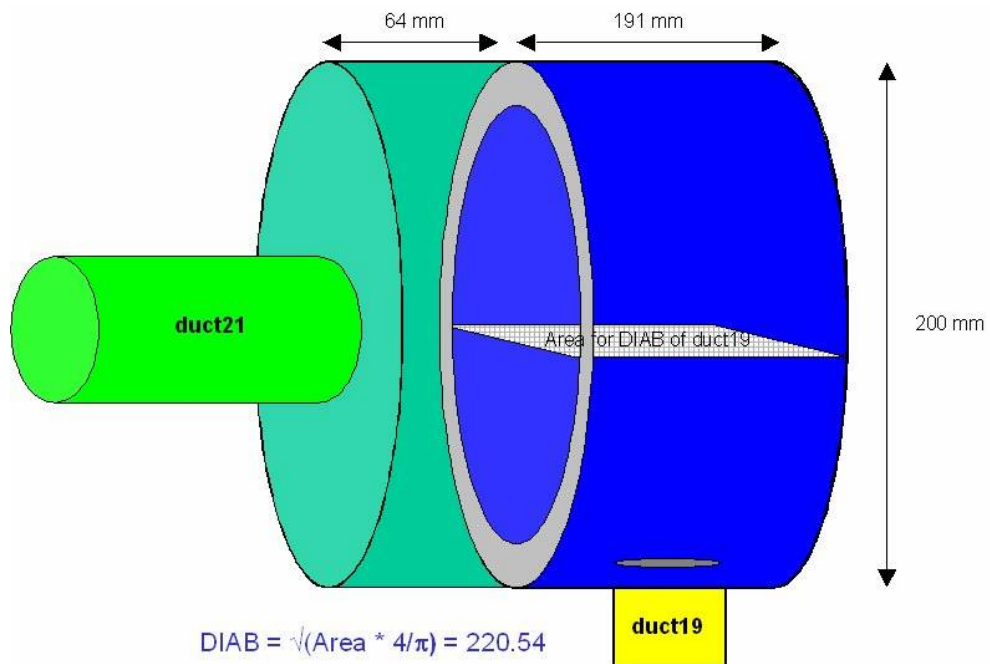


Figure 5: DIAB Value for duct19

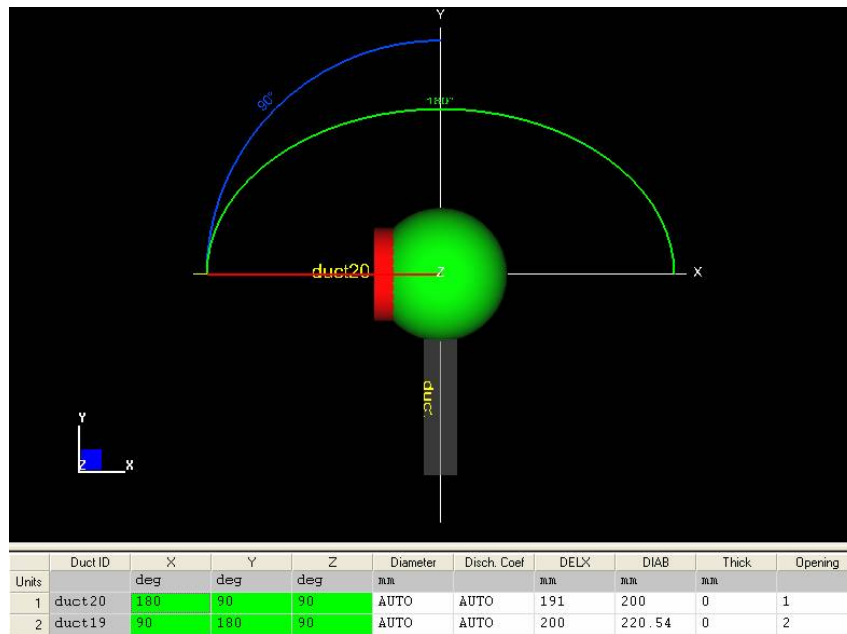


Figure 6: Orientation Panel for 1st Complex Y- Junction

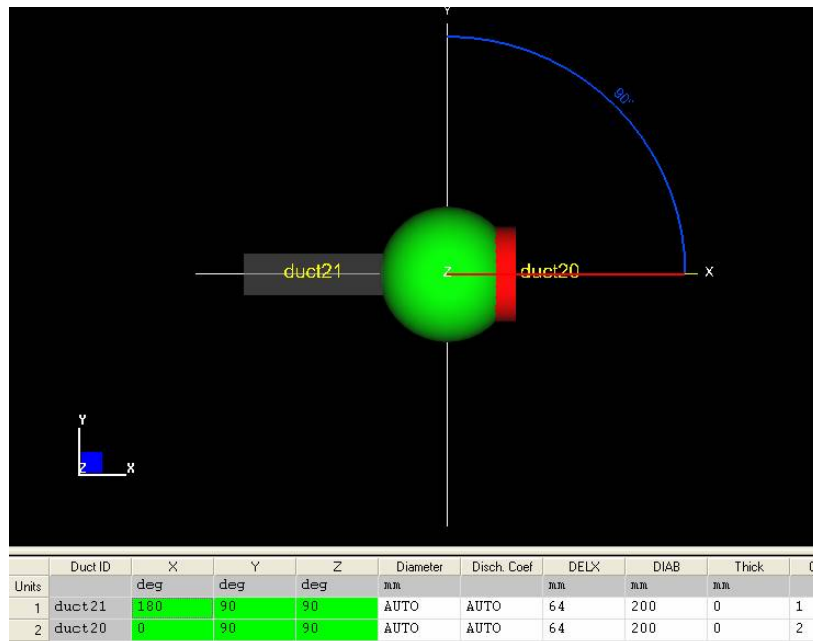


Figure 7: Orientation Panel for 2nd Complex Y- Junction

## 2.2 Representing the Filter as a Massless Duct

The filter will be modeled as a massless duct (overall length of 0) and will act like an orifice plate between the two volumes by having a smaller diameter than the surrounding volumes.

Typical information provided for an air filter is a pressure drop achieved at a given mass flow rate (most frequently, near the maximum expected mass flow rate). For our air filter, we are supplied with the information the filter creates a **1.25 kPa** pressure drop across the system at a flow rate of **280 kg/hr**.

The Diameter of **duct20** can be obtained by steady-state flow simulation with a fixed pressure drop by putting ambient junctions at either end of the system and running a 1second time-based simulation. Varying the orifice diameter will produce different mass flow rates and we can target the desired flow rate with the prescribed pressure drop. To understand this setup, see: [.\\Ricardo\\WAVE\\7.0\\examples\\engine\\TUT\\_sl\\Air\\_Filter\\_Calibration](#)

In this case, the orifice diameter of **140.37 (mm)** provides for a 1.25 kPa pressure drop at 280 kg/hr mass flow of air. Massless ducts need only have **Left and Right Diameters** defined (these must be equal!) and the **Overall Length** should be set to **0**. No other parameters need be set. When completed, the massless Duct Panel should appear as in Figure 8.

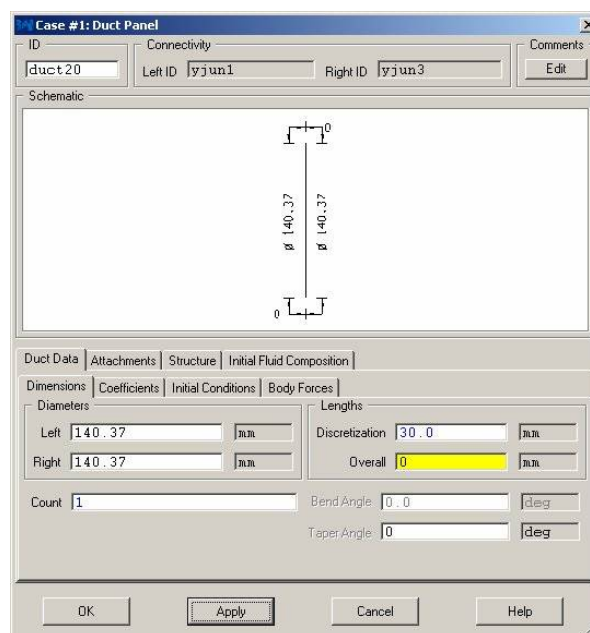


Figure 8: Massless Duct Panel

**Save your model**

### 3. The Throttle Body

In this step, we will add a very simple, parameterized representation of a throttle body. Although this Tutorial won't do so, this common practice allows for the engine load to be varied between cases by changing the open area of the throttle body.

#### 3.1 The Basic Geometry

The schematic of the throttle body geometry is shown in **Figure 1**. In WaveBuild, we will model this geometry as two ducts connected in the middle using a throttle valve junction. The throttle valve junction will represent the throttle blade.

Place the required junctions on the canvas and connect with ducts as shown in **Figure 2**, starting with the final orifice junction of the zip tube. Edit both ducts to have **Overall Lengths of 50 (mm)** and constant **Diameters of 60 (mm)**. Remember to set **the Discretization Length to 35 (mm)** and the **Initial Conditions** to the standard of 1 bar **Pressure and 300 K Initial Fluid and Wall Temperatures**.



Figure 1: Throttle Body Schematic Figure 2: WaveBuild Throttle Body

ID	Connectivity	
duct23a24	Left ID orif11	Right ID THROTTLE
Schematic		
Duct Data   Attachments   Structure   Initial Fluid Composition		
Dimensions   Coefficients   Initial Conditions   Body Forces		
Diameters Left 60 mm Right 60 mm		Lengths Discretization 35 mm Overall 50 mm
Count 1		Bend Angle 0.0 deg Taper Angle 0 deg

#### 3.2 Defining the Butterfly Valve

The throttle valve junction represents an instance of a butterfly valve type, as defined in the Model -> Valves... list. In order to use it, we must first define the butterfly valve. Click on the **Valves...** menu item in the **Model pull-down menu**. Click on the **Add** button in the Valve List and select the **Butterfly** valve type, then click on the **OK** button to open the Butterfly Valve Editor panel. Set the Bore Diameter to **60 mm** and the Shaft Diameter to **5 mm**, with a Minimum Plate Angle (angle at which the valve sits effectively closed) of **5 deg**. The Calculated Values section of the panel will allow you to preview the calculated geometric quantities. Use the slider bar to observe the schematic and values as the butterfly valve angle changes. When completed, the Butterfly Valve Editor panel should appear as in **Figure 3**.

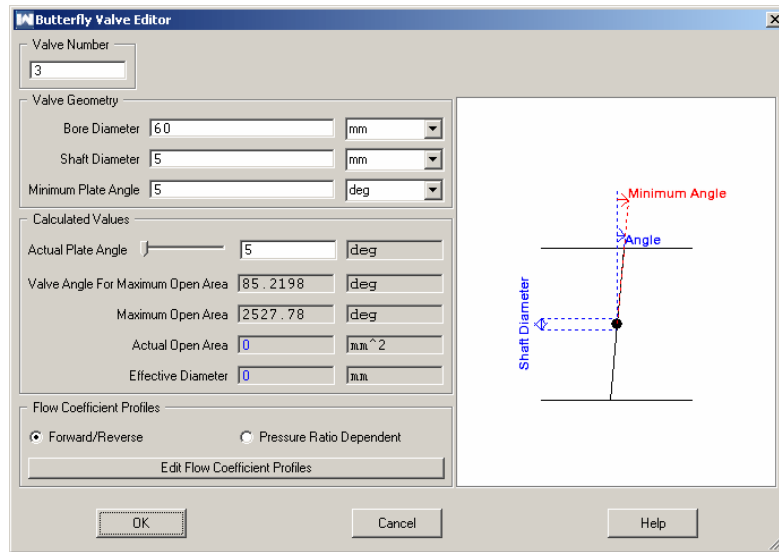


Figure 3: Parameterizing the Throttle Blade Setting

With the **Forward/Reverse** coefficient profile type radio button selected, click on the **Edit Flow Coefficient Profiles** button to open the flow coefficient profile editor panel. If you have a defined profile as a function of angle, it should be entered here. For the purposes of this simulation we will define a coefficient of **0.5 at 5 deg** and **1 at 85.22 deg**, allowing a full sweep of realistic coefficient values from when the valve is closed until it is fully open. When completed, the flow coefficient profile editor should appear as in **Figure 4**.

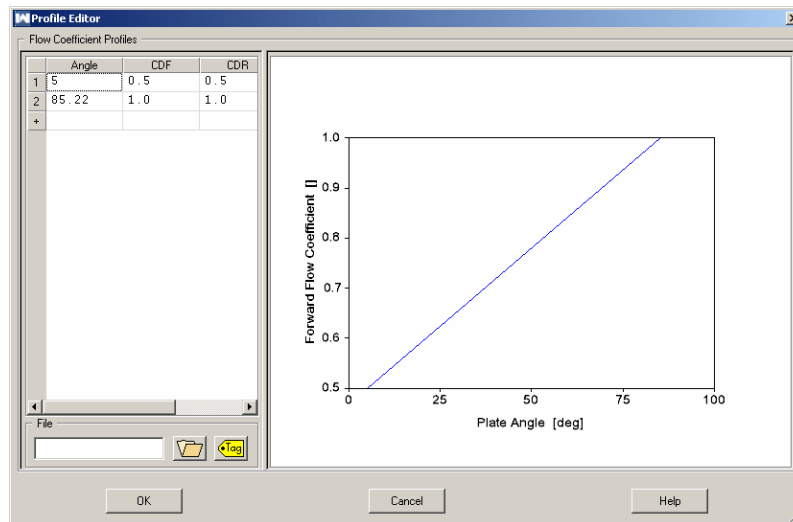


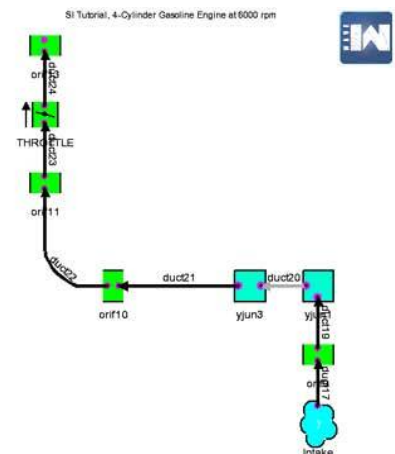
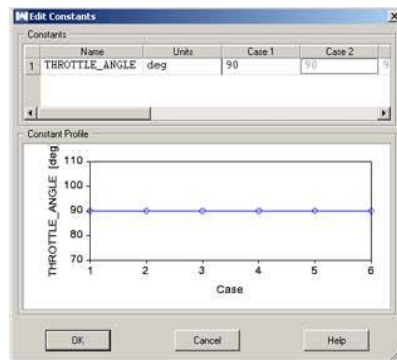
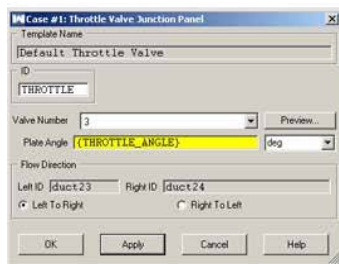
Figure 4: Setting the throttle constant

### 3.3 Using the Throttle Valve Junction

**Double-click on the throttle valve** junction on the canvas to open the Throttle Valve Junction Panel and edit the **ID** of the junction to **THROTTLE**. Select **Valve Number 3** from the pull-down list and set the **Plate Angle to {THROTTLE\_ANGLE}** (see **Figure 5**). This parameterizes the throttle plate angle, allowing us to set the angle in the Constants Panel, so it can change between cases, giving load control during the simulation.

The background of the **Plate Angle** field will turn yellow as the constant **THROTTLE\_ANGLE** is not yet defined. Click the **OK** button to save the settings and close the panel and, when queried about adding the constant to the constants table, click the OK button again and set the value **90** for all cases (setting it in **Case #1** only is sufficient, see **Figure 6**). Modeling the throttle fully open assumes we are running a full-load speed sweep.

When completed, the intake subsystem should appear as in **Figure 7** (below).

 Save your model

#### 4. The Intake Manifold

## 4.1 The Basic Geometry

The schematic of the intake manifold geometry is shown in **Figure 1**. In WaveBuild, we will model the inlet pipe as a single duct. The plenum will be simplified and assumed cylindrical in shape (with an equivalent diameter of 110 mm). Four Complex Y-junctions will be attached together using massless ducts to represent the plenum volume. Three ducts will be used to represent each intake runner. A discharge coefficient will be imposed for incoming flow to the first runner duct, to model the bellmouth entry.

The equivalent network as created in WaveBuild is shown in Figure 2. Use the **Junction Palette...** to place the required **junctions** and connect them with **ducts** accordingly. Remember to follow the **Left to Right convention**. Start the inlet pipe of the intake manifold at the last **orifice junction** of the throttle body. Use four **Complex Y-junctions** to model the intake manifold plenum. These four Y-junctions will be connected using massless ducts (zero overall length).

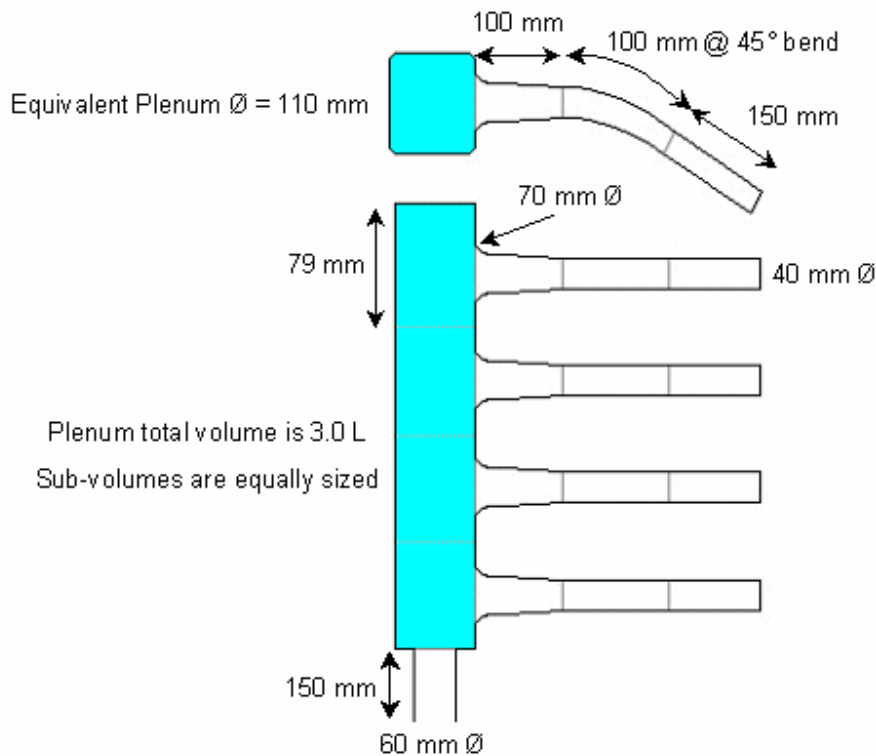


Figure 1: Intake Manifold Schematic

## 4.2 Modeling the Plenum Sub-Volumes

Edit the duct representing the inlet to the intake manifold. Assign **Left** and **Right Diameters** of **60 (mm)** and an **Overall Length** of **150 (mm)**. Remember to set the **Discretization Length** to **35 (mm)** and the **Initial Conditions** to **1.0 bar** Pressure and **300 K** Initial **Fluid** and **Wall Temperatures**.

The **Left** and **Right Diameters** of the massless ducts should equal the equivalent diameter of the Y-junctions (which will also match the **DIAB** values assigned to the connections) so that there is no pressure loss due to expansion or contraction. The Y-junctions have an equivalent diameter of **110 mm**, so assign **Left** and **Right Diameters** of **110 (mm)** to the massless ducts, along with **0 (mm) Overall Lengths** (see **Figure 3**). They will appear gray on the canvas, indicating they are massless ducts.

The Y-junctions should be edited to have **Diameter** values of **110 (mm)**, **Volume** values of **0.75e+006 (mm<sup>3</sup>)** (3.0 L divided evenly by four), and **Heat Transfer/Skin Friction Area** values of approximately **27300 (mm<sup>2</sup>)** ( $\pi \times \text{diameter} \times \text{length}$ ) as in **Figure 4**.

Orient the duct connections according to the layout on the screen (see **Figure 2**). The **DELX** values for the massless duct connections should be **79 (mm)** (the length of each subvolume in the direction of flow through the massless ducts) and the **DIAB** values should be **110 (mm)** (equal to the massless duct Diameters so no expansion or contraction occurs).

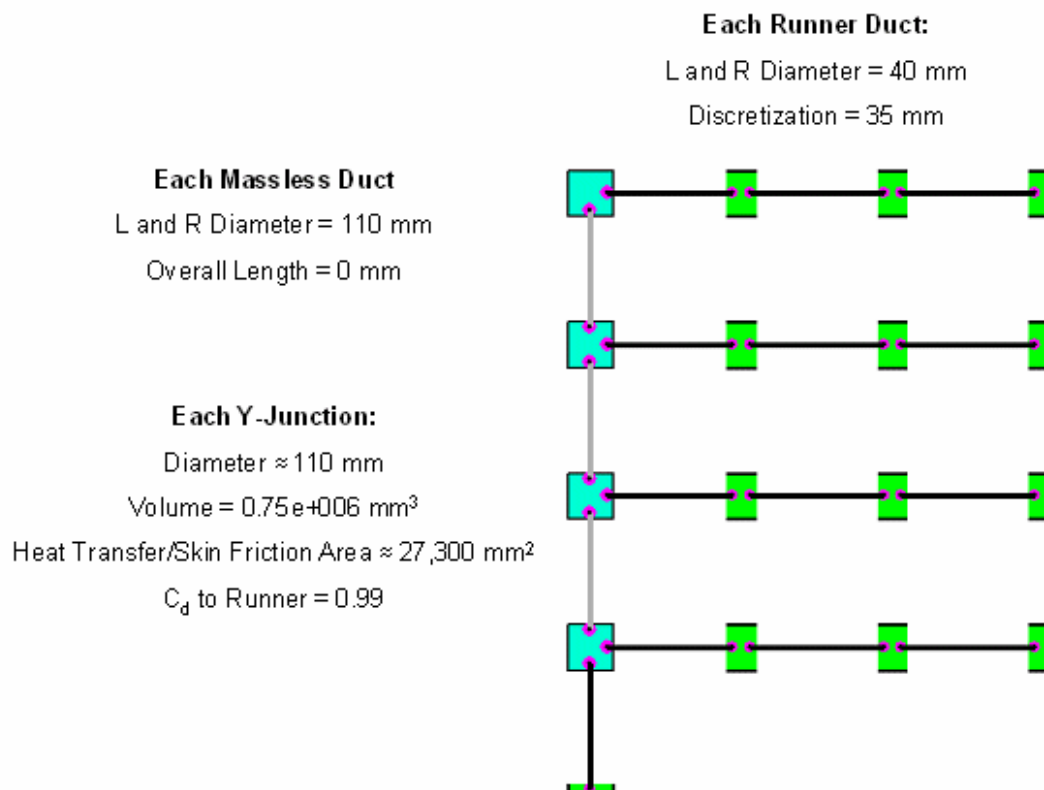


Figure 2: WaveBuild Intake Manifold

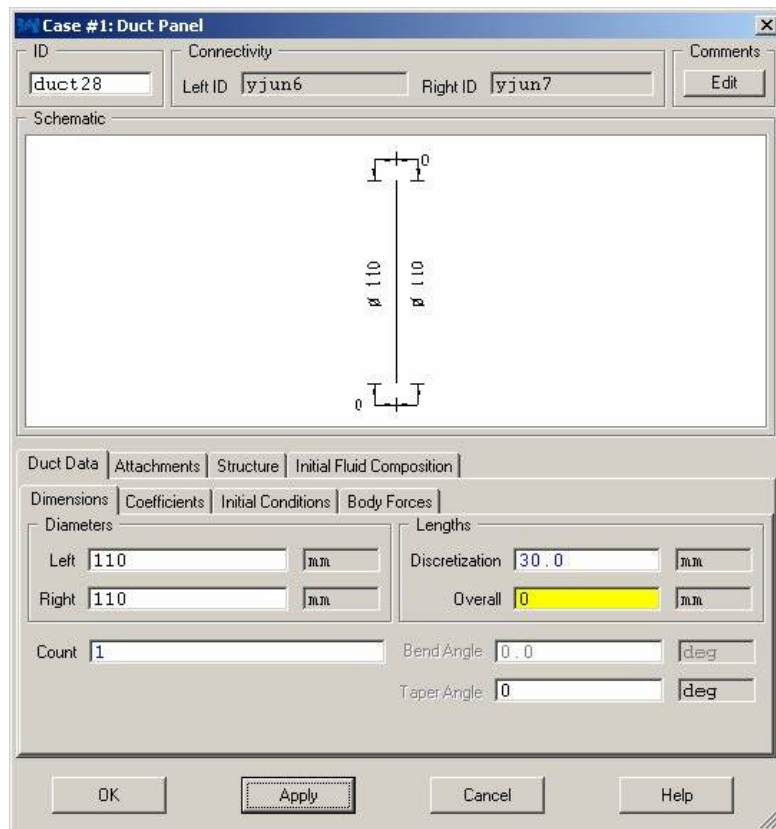


Figure 3: Massless Duct between Plenum Sub-Volumes

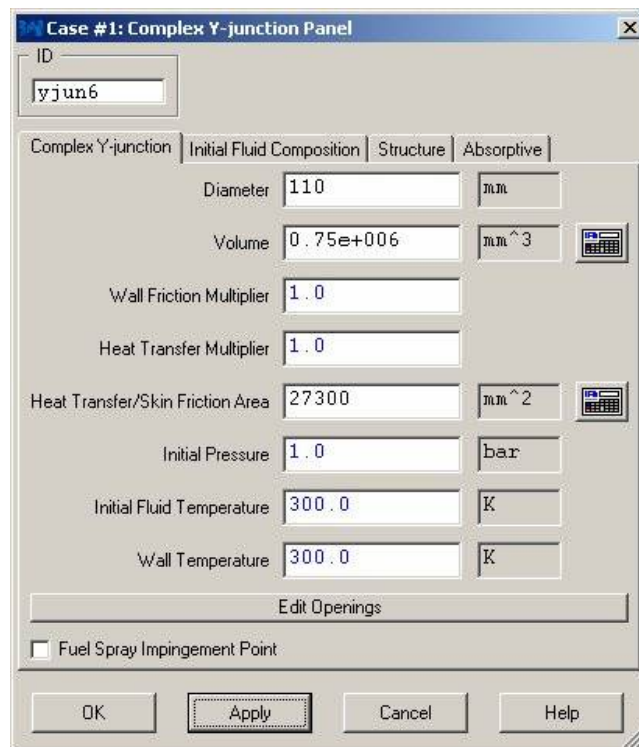


Figure 4: Y-Junctions Representing Plenum Sub-Volumes



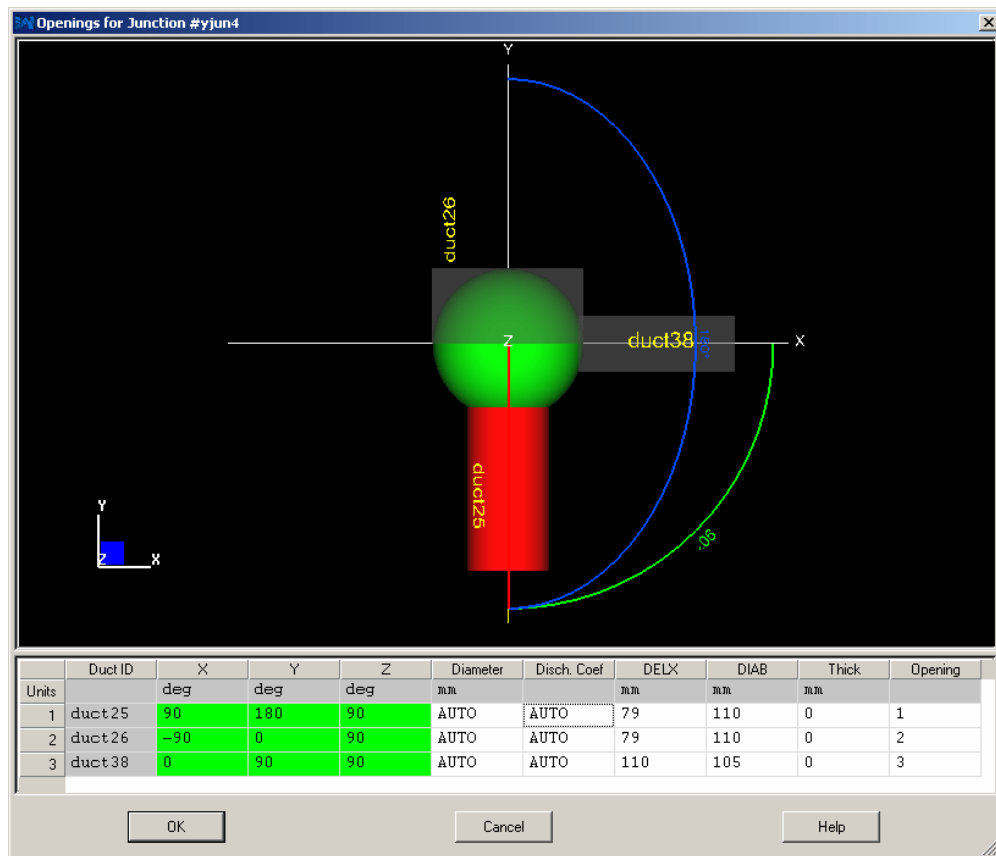


Figure 5: Duct Orientation for First Plenum Sub-Volume

The **DELX** values for the runner connections should be **110 (mm)** (distance across the subvolume in the direction of flow into the runners). The **DIAB** values for the runner connections require some thought. Should the area used to calculate the **DIAB** value for each runner connection be the maximum area the gas can expand into in the Y-junction or the length of the entire plenum?

Technically, the **DIAB** value should be calculated from the maximum area the gas can expand into *along the length of the single y-junction into which the duct enters*. This is because any losses caused by flow traveling along the length of the plenum will be accounted for by mass transfer from one y-junction to the next. For our geometry, the maximum area for expansion in the direction of flow from the runners is equal to  $110 \text{ mm} * 79 \text{ mm} = 8690 \text{ mm}^2$ . Thus **DIAB** is approximately **105 (mm)**. See **Figure 5** for a representative duct orientation.

Note, the pressure and flow after a sudden expansion has a greatly diminished response the larger the expansion is. Once the **DIAB** value is approximately twice the diameter of the entering duct, the effect of the expansion changes very little with further increase in **DIAB**. **Figure 6** illustrates this effect.

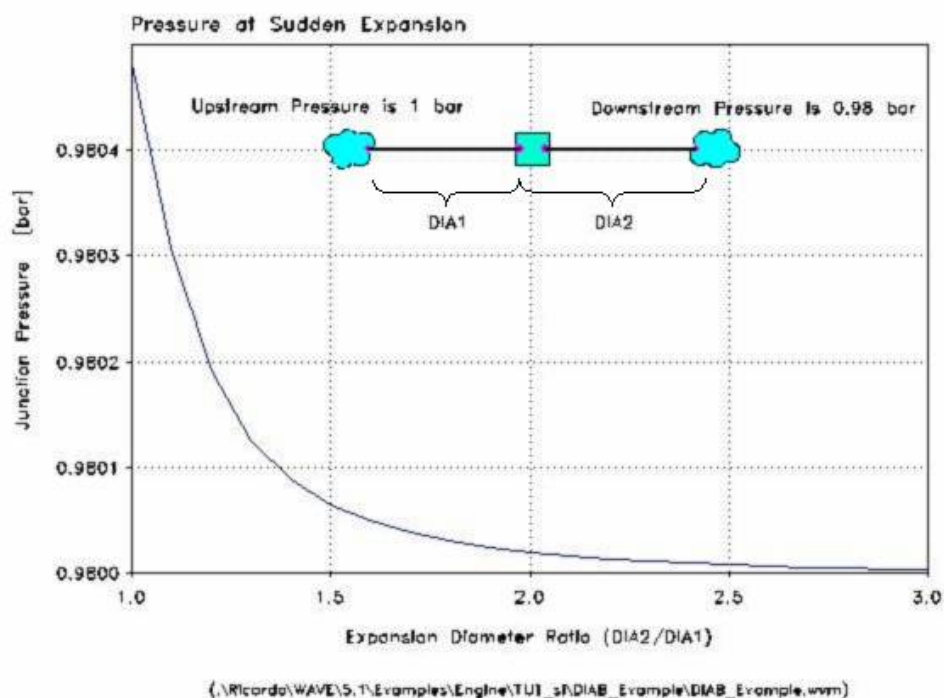


Figure 6: DIAB Sensitivity Illustration

To edit multiple ducts at the same time (junctions can also be edited in this manner, in combination with ducts if desired), multiple-select the items to be edited by holding the shift key and left-clicking on the items or by drawing a box around the desired items while holding down the left mouse button. With the desired items selected (highlighted in red) right-click on the white background of the canvas and select the **Edit Parameters...** menu option. This will open the Parameters Panel allowing fields to be set for multiple ducts/junctions simultaneously (see Figure 7 for example of setting duct geometry for all runner ducts).

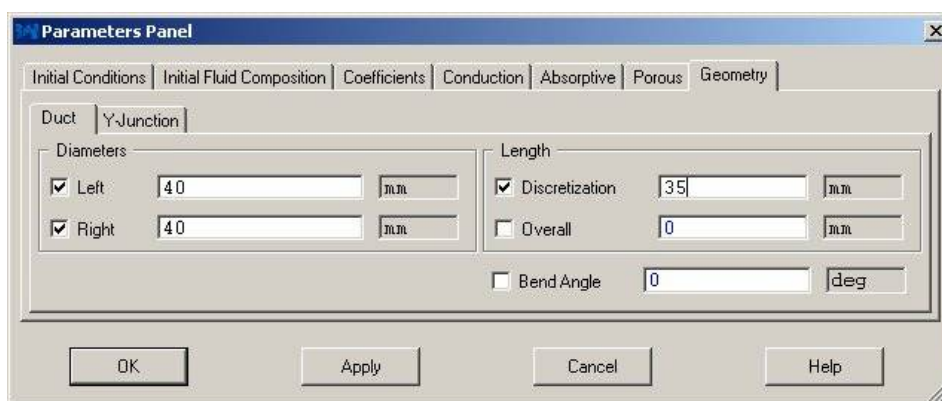


Figure 7: Parameters Panel

This means that during reverse flow, from the runner duct to the plenum Y-junction, it will experience a sudden expansion into the junction and the loss will be accounted for. To accurately represent the effect of the bellmouth entry, the Disch. Coef. from the plenum junction to the runner duct should be set to a representative high value, in the range of 0.95 - 1.0 in the Openings panel for each Y-junction (see Figure 8).

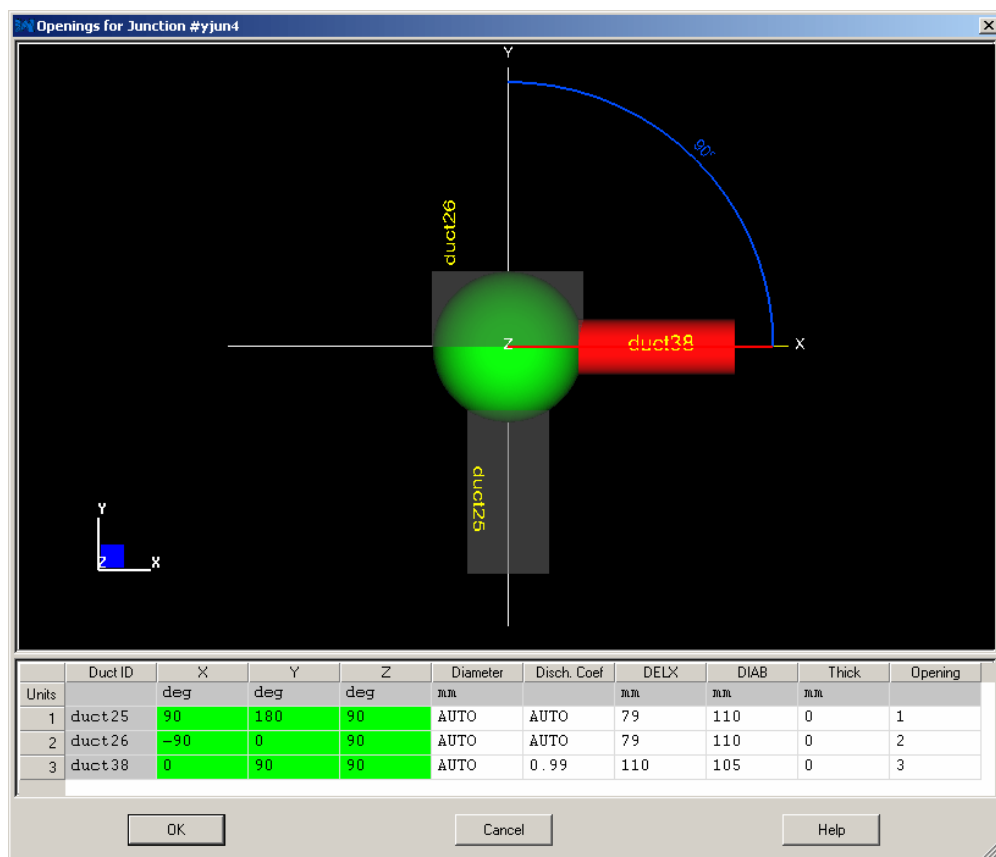
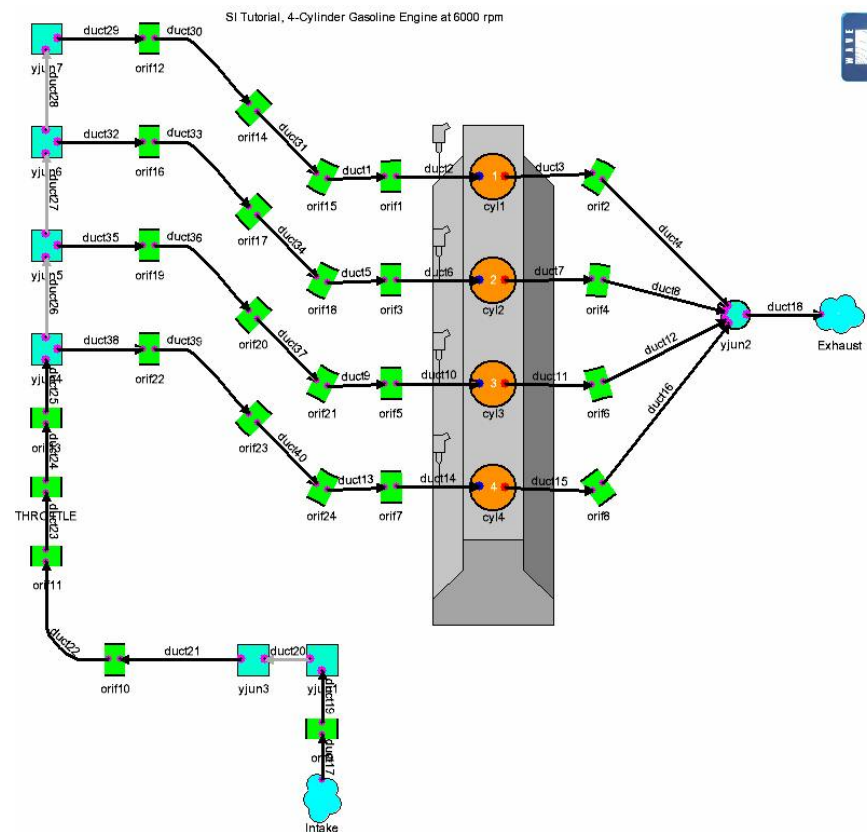


Figure 8: Setting the Discharge Coefficient to 0.99



duct 29, 32, 35, 38:

duct29	Left ID   yjun7	Right ID   orif12
Schematic		
Duct Data   Attachments   Structure   Initial Fluid Composition		
Dimensions   Coefficients   Initial Conditions   Body Forces		
Diameters		
Left	40	mm
Right	40	mm
Lengths		
Discretization	35	mm
Overall	100	mm
Count	1	
Bend Angle	0.0	deg
Taper Angle	0	deg

duct 30, 33, 36, 39:

ID	Connectivity	
duct30	Left ID   orif12	Right ID   orif14
Schematic		
Duct Data   Attachments   Structure   Initial Fluid Composition		
Dimensions   Coefficients   Initial Conditions   Body Forces		
Diameters		
Left	40	mm
Right	40	mm
Lengths		
Discretization	35	mm
Overall	100	mm
Count	1	
Bend Angle	45	deg
Taper Angle	0	deg

duct 31, 34, 37, 40:

ID		Connectivity	
duct31	Left ID	orif14	Right ID
Schematic			
Duct Data   Attachments   Structure   Initial Fluid Composition			
Dimensions   Coefficients   Initial Conditions   Body Forces			
Diameters		Lengths	
Left	40 mm	Discretization	35 mm
Right	40 mm	Overall	150 mm
Count	1	Bend Angle	0.0 deg
		Taper Angle	0 deg

duct 25:

ID		Connectivity	
duct25	Left ID	orif13	Right ID
Schematic			
Duct Data   Attachments   Structure   Initial Fluid Composition			
Dimensions   Coefficients   Initial Conditions   Body Forces			
Diameters		Lengths	
Left	60 mm	Discretization	35 mm
Right	60 mm	Overall	150 mm
Count	1	Bend Angle	0.0 deg
		Taper Angle	0 deg

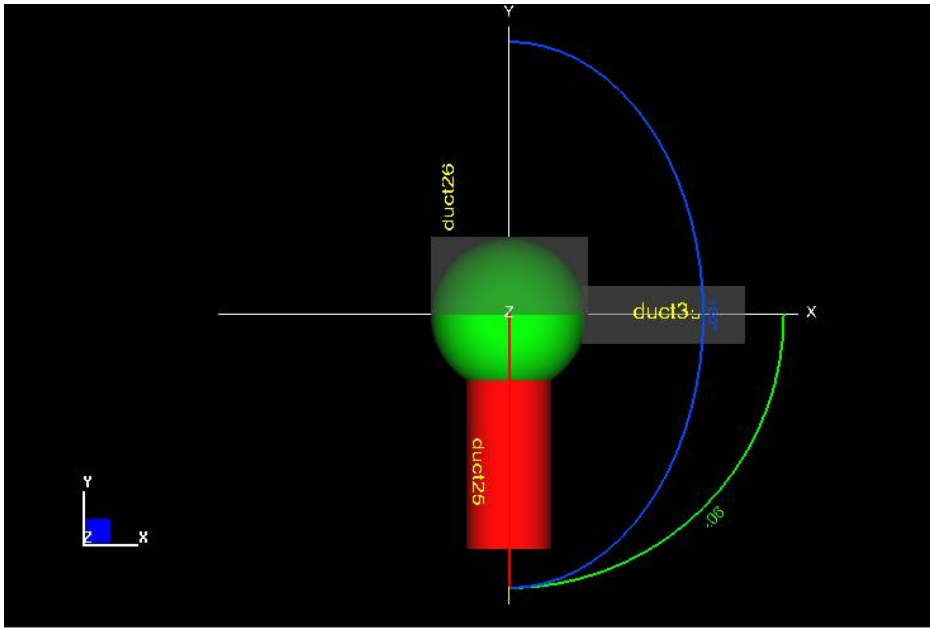
duct 26, 27, 28:

ID		Connectivity	
duct26	Left ID	yjun4	Right ID
Schematic			
Duct Data   Attachments   Structure   Initial Fluid Composition			
Dimensions   Coefficients   Initial Conditions   Body Forces			
Diameters		Lengths	
Left	110 mm	Discretization	35 mm
Right	110 mm	Overall	0 mm
Count	1	Bend Angle	0.0 deg
		Taper Angle	0 deg

yjun 4, 5, 6, 7:

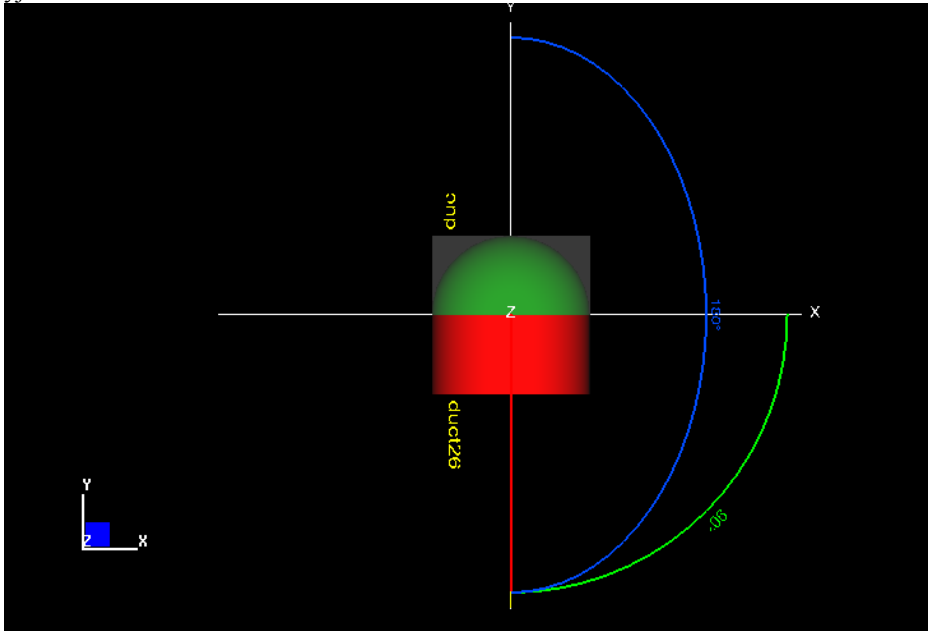
ID	
yjun6	
Complex Y-junction   Initial Fluid Composition   Structure   Absorptive	
Diameter	110 mm
Volume	0.75e6 mm^3
Wall Friction Multiplier	1.0
Heat Transfer Multiplier	1.0
Heat Transfer/Skin Friction Area	27300 mm^2
Initial Pressure	1.0 bar
Initial Fluid Temperature	300.0 K
Wall Temperature	300.0 K
Edit Openings	
<input type="checkbox"/> Fuel Spray Impingement Point	

yjun 4: ( $DIAB = \sqrt{Area \cdot \frac{4}{\pi}} = \sqrt{110 \cdot 79 \cdot \frac{4}{\pi}} \cong 105mm$ )



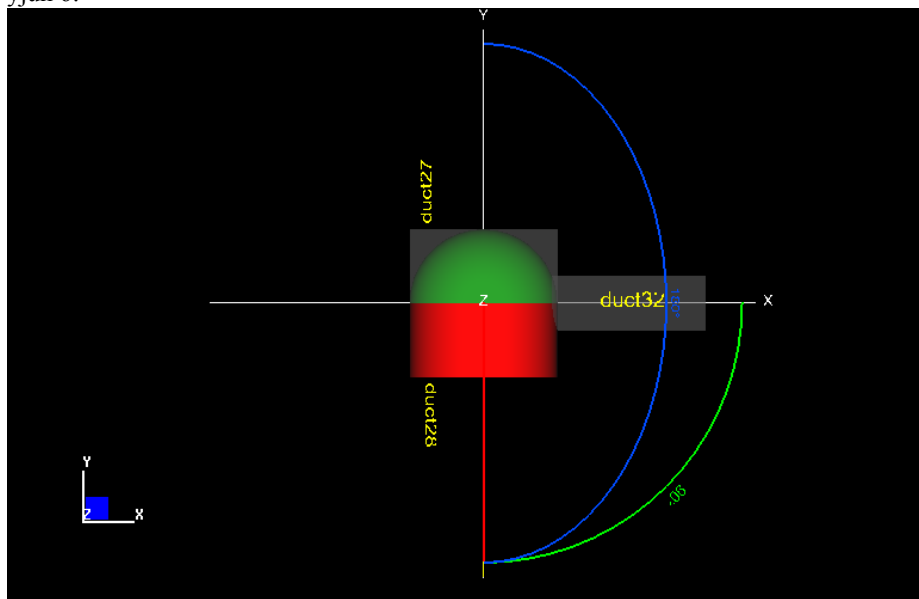
	Duct ID	X	Y	Z	Diameter	Disch. Coef	DELX	DIAB	Thick	
Units		deg	deg	deg	mm		mm	mm	mm	
1	duct25	90	180	90	AUTO	AUTO	79	110	0	1
2	duct26	-90	0	90	AUTO	AUTO	79	110	0	2
3	duct38	0	90	90	AUTO	0.99	110	105	0	3

yjun 5:



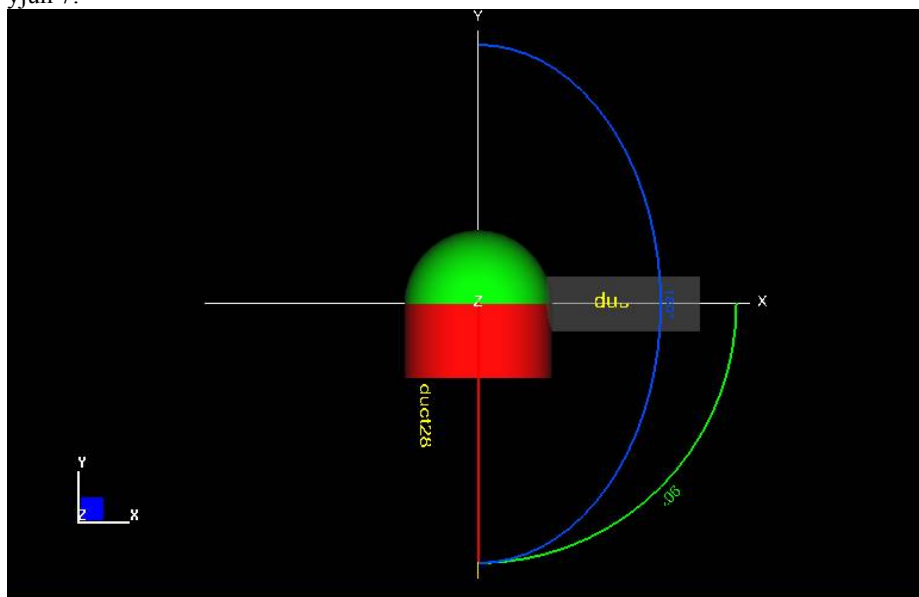
	Duct ID	X	Y	Z	Diameter	Disch. Coef	DELX	DIAB	Thick	
Units		deg	deg	deg	mm		mm	mm	mm	
1	duct26	90	180	90	AUTO	AUTO	79	110	0	1
2	duct27	-90	0	90	AUTO	AUTO	79	110	0	2
3	duct35	0	90	90	AUTO	0.99	110	105	0	3

yjun 6:



	Duct ID	X	Y	Z	Diameter	Disch. Coef	DELX	DIAB	Thick	Opening
Units		deg	deg	deg	mm		mm	mm	mm	
1	duct28	90	180	90	AUTO	AUTO	79	110	0	1
2	duct27	-90	0	90	AUTO	AUTO	79	110	0	2
3	duct32	0	90	90	AUTO	0.99	110	105	0	3

yjun 7:



	Duct ID	X	Y	Z	Diameter	Disch. Coef	DELX	DIAB	Thick	Opening
Units		deg	deg	deg	mm		mm	mm	mm	
1	duct28	90	180	90	AUTO	AUTO	79	110	0	1
2	duct29	0	90	90	AUTO	0.99	110	105	0	2


 Save your model

## 5. Creating Animations in WavePost

WavePost can use time-based data to animate the network with color contours of basic datasets. These animations can be viewed in WavePost and then saved as MPEG files for use in presentations and reports.

### 5.1 Requesting Animation Data in WaveBuild

In WaveBuild, open the Output and Plotting Panel by selecting the **Output and Plotting...** item from the **Simulation pull-down menu**. Check the **Generate Animation Data** checkbox and click **OK** to save the setting and close the panel (see **Figure 1**).

Run the WAVE model by clicking on the **Run Screen Mode** button in the toolbar  (if prompted to save the model before running WAVE, click the **OK** button to save and run sequentially).

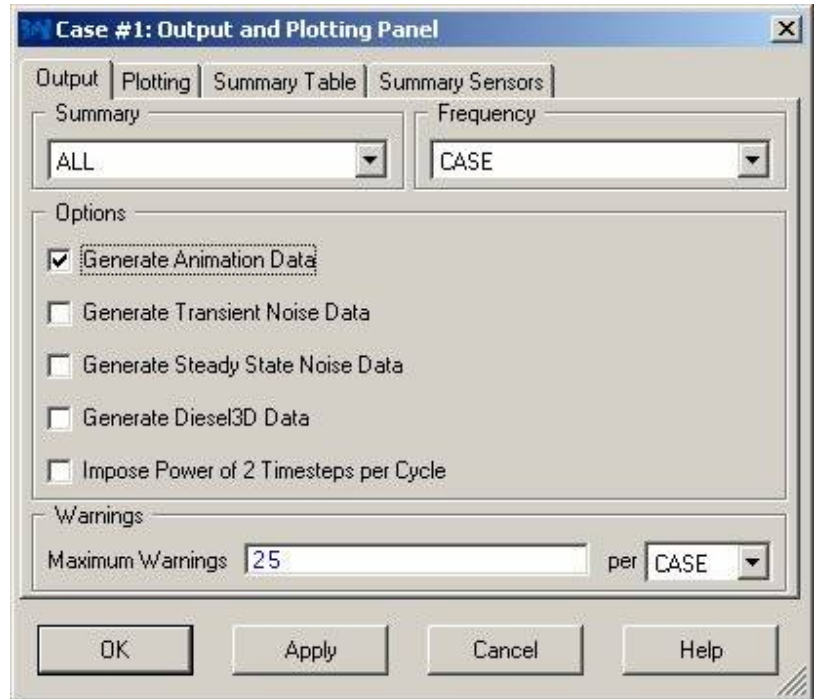



Figure 1: Requesting Animation Data

### 5.2 Adding Results to an Existing .wps File

left-clicking in the Launch WavePost button in the WaveBuild toolbar - 

Click on the **Add** button at the bottom of the **Output Files** frame and select the **newly created .wvd file** for the WAVE model containing the intake system. The .wvd and .sum files for this WAVE run will be added to the Output Files list (see **Figure 2** below). A Query window will pop-up prompting whether to add curves to the existing plots using this file (see **Figure 3**). Click on the **Yes** button and every existing plot will add data from the newly added .wvd and .sum files (if matching data exists in the new files). Open the **Sweep Plots** to view the comparison of the performance parameters as in **Figure 4**.

Note that addition of the intake system has changed the predicted performance results. Power is decreased near 3000 rpm, but increased above 4000 rpm. The positive tuning effects are more powerful than the losses due to friction, expansion and contraction, and bends that were added in the intake system.





Figure 2: WavePost Output Files Frame

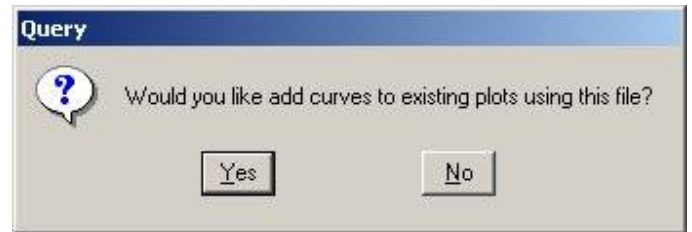


Figure 3: Query to Add Data to Existing Plots

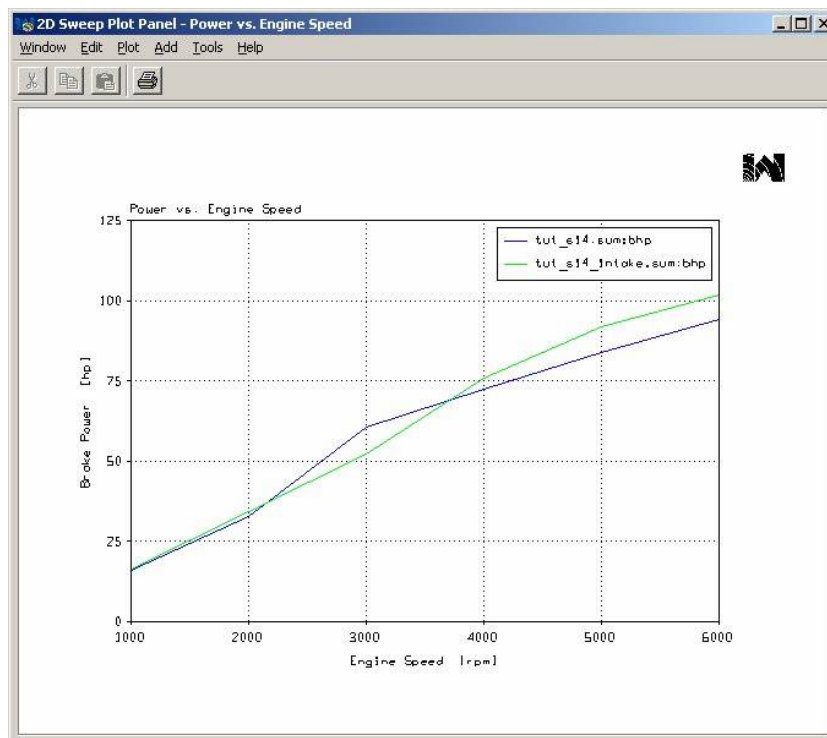
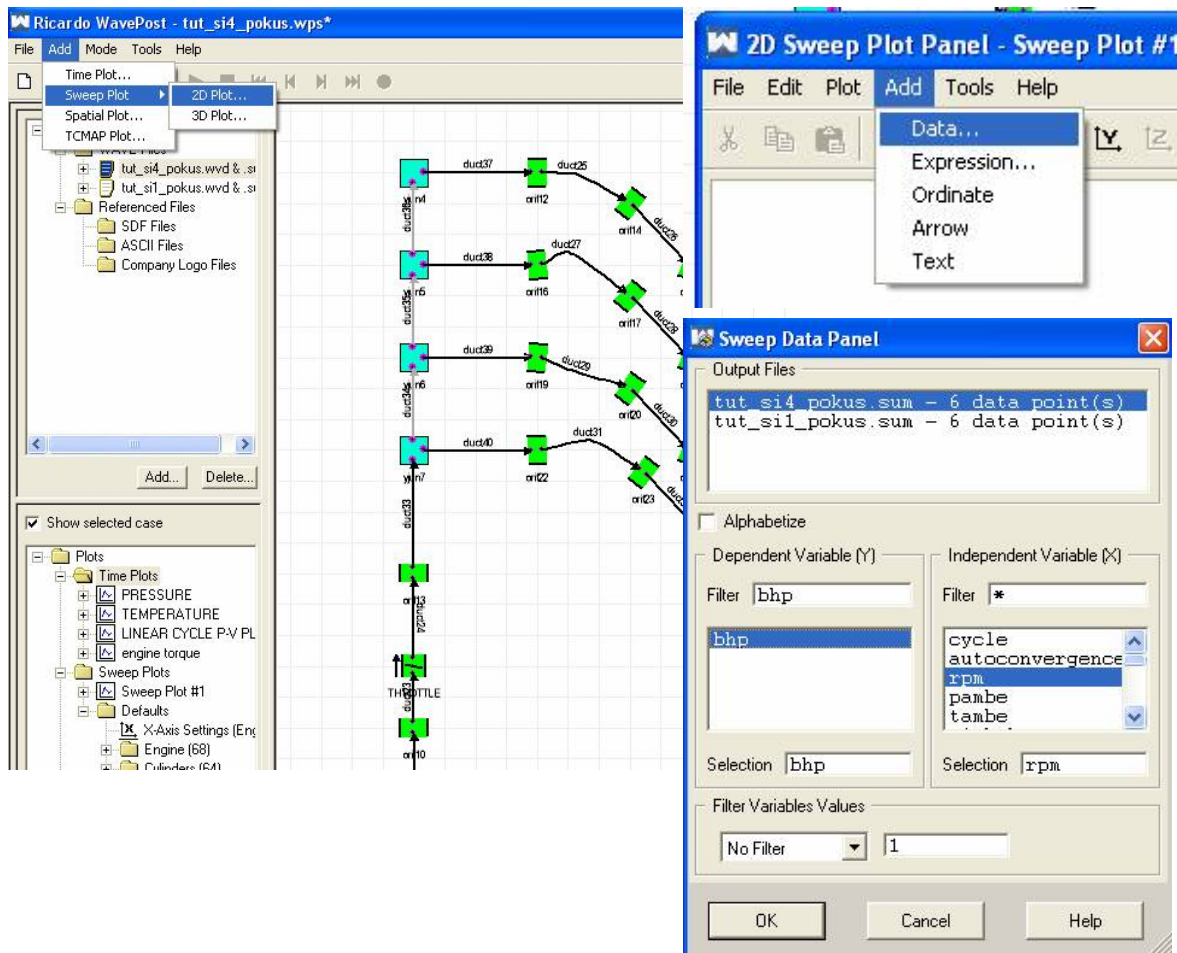


Figure 4: Data Automatically Added to Power Plot



### 5.3 Creating and Saving Animations in .mpg Format

Animations can be created using time data stored in the .wvd file. Select the **Animation** option from the **Mode pull-down** menu. The main canvas will appear similar to the Cycle-Averaged display, with ducts and junctions scaled to their relative diameters and colored according to the currently-selected dataset and contour levels.

Under the **Tools pull-down** menu, select the **Variable...** option to open the Variable Panel. In the Variable Panel, the data being animated can be selected and the appropriate range can be set. Under the Variables list, select the **Velocity** dataset. Set the Variable Range Max to **180 (m/s)** and set the Min to **-20 (m/s)** as in **Figure 5** (type the values and press the Enter key).

Also under the **Tools pull-down** menu, select the **Display** option to open the Display Panel. In the Display Panel, certain aspects of the canvas display can be altered in appearance. Click the Crank Animation checkbox to display a cylinder crank animation on the main canvas, in the upper right-hand corner. This animation displays the position of the piston in cylinder one and colors the combustion chamber appropriately to the contour selected. Move the Contours slider bar all the way to the left to make the smoothest **contour** band possible. Also set the Number of Interval **Labels** to **11** and hit Enter to label the contour band on the canvas evenly. The Display Panel should appear as in **Figure 6**.

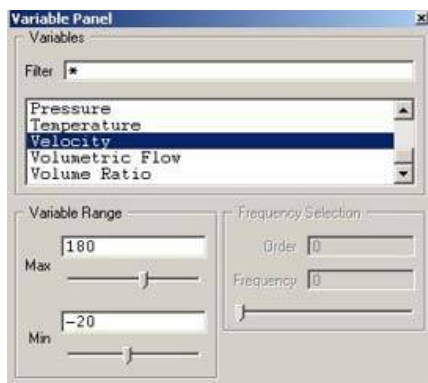


Figure 5: Velocity Dataset and Contour Range

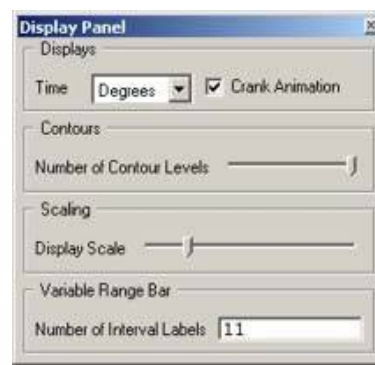


Figure 6: The Display Panel

Again, under the **Tools pull-down** menu, select the **Time** option to open the Time Panel. The Time Panel allows the user to select how many frames will be made in the animation (or how large a step each frame will be). Each input field in the Time Panel recalculates the values in the other input fields (i.e. they are all related to each other). Set the Delta for the Degr field to **2** and hit the Enter key. This should create **359** frames in the animation, each frame being a 2° crank angle step in the engine cycle. The Time Panel should appear as in **Figure 7** below.

Finally, under the **Tools pull-down** menu, select the **Animation...** option to open the Animation Panel. This panel allows playback of the animation, stepping through to particular locations, and recording of the animation to **MPEG format** (alternatively, these functions can be accessed by the toolbar buttons above the canvas). **Left-click** on the **record button** to open the Movie Recordings Settings Panel. Enter a name for the movie in the File text field (**Velocity.mpg**) and set the MPEG Quality to **High** (this will use more disk space but create a clearer image) as in **Figure 8**. Click on the **OK** button and a progress bar will pop up to show the progress of the rendering process. This will take a while to complete.

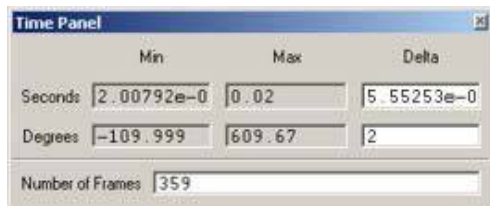


Figure 7: The Time Panel with 2° Steps



Figure 8: The Movie Recordings Settings Panel

When finished, a file with the name as given above will be created in the working directory. It should appear similar to the animation below (right-click on the animation and select Play to repeat).

 **Save your file**