GT-SUITE

Engine Block Thermal Modeling Tutorial

VERSION **2017**

by

Gamma Technologies

GTI SUPPORT

- TELEPHONE: (630) 325-5848
- FAX: (630) 325-5849
- E-MAIL: support@gtisoft.com
- Web Address: www.gtisoft.com
- Address: 601 Oakmont Lane, Suite 220
  Westmont, IL 60559
  USA

Telephone Support Hours

8:00 A.M. to 5:30 P.M. Central Time Monday – Friday

TABLE OF CONTENTS

TUTORIAL 1: Engine Block Thermal Modeling 1

1.1 Required information 1

1.1.1 Geometry 1

1.1.2 CFD - Calibration Data 1

1.1.3 Engine model 2

1.1.4 Friction Model 2

1.2 Water Jacket Modeling 2

1.2.1 Extracting the Water Jacket Volumes 3

1.2.2 Converting the Water Jackets in GEM3D 12

1.2.3 Adding External Connections 31

1.3 Creating Cylinder Structure Parts in GEM3D 32

1.3.1 EngCylStrucCond Dimensions 32

1.3.2 Creating the EngCylStrucCond Component 37

1.3.3 Water Jacket Definitions 49

1.3.4 Valve and Port Definitions 50

1.3.5 Creating Other Cylinders 52

1.4 Creating Thermal Masses for the Engine Block and Head 54

1.4.1 Converting the Thermal Masses 54

1.4.2 Adding Ports to the Thermal Masses 56

1.4.3 Adding Convection Connection Between Block and Water Jacket 62

1.5 Discretizing the GEM3D model 63

1.6 Linking the Model in GT-ISE 65

1.6.1 Connecting Water Jackets to the EngCylStrucCond 66

1.6.2 Connecting Water Jackets to the Structure masses 70
1.6.3 Connecting the Block to the Head 72
1.6.4 Connecting the EngCylStrucCond to the Structure 73
1.6.5 Creating Oil Volumes in the Block and Head 75
1.7 Calibrating the Model 78
1.7.1 Required data for calibration 78
1.7.2 Calibrating the model pressure loss 79
1.7.3 Imposing the heat transfer coefficients 81
1.7.4 Imposing the engine boundary conditions 82
1.7.5 Model Setup 85
1.7.6 Calibration Guidelines 87
1.7.7 Preparing the Model for Transient Operation 89
1.8 Integration with a Fast-Running Engine Model 90
1.8.1 Connecting the Models 90
1.8.2 Run Setup for Multiple Circuits 95
1.8.3 Coloring a Circuit (Optional) 97
1.9 Integration with Other Circuits 98
1.9.1 Integrating the Coolant System 98
1.9.2 Integrating the Oil System 99
1.9.3 Integrating the Underhood System 100
1.10 Friction Modeling for Thermal Warmup 101
1.10.1 Using Friction from Strip-Down Tests 101
1.10.2 Using Theoretical Models 102
TUTORIAL 2: Definition and Use of 'EngCylStrucCustom' in GT-SUITE v2016 103

2.1 Steps for Building the EngCylStrucCustom in GT-SUITE v2016 103
2.1.1 Import the CAD geometry to GEM3D 103
2.1.2 Convert the imported geometry to a Finite Element component 103
2.1.3 Create the surface ports on the GEMThermalFE component 104
2.1.4 Create the .gtm file 106
2.1.5 Create a new EngCylStrucCustom object 107
2.1.6 Create an FECylStrucCustom object 107
2.1.7 Test the Structure 114

TUTORIAL 1: Engine Block Thermal Modeling

This tutorial has been prepared to assist users of GT-SUITE in building thermal models of engine systems. This tutorial assumes that a CAD model of the engine block and related systems are available; sample geometry is provided. The tutorial will guide you through preparing the geometry in GT-SPACECLAIM, using GEM3D to import the geometry and convert it to GT-SUITE parts, connecting the thermal and flow systems in GT-ISE, calibrating the model, and imposing a friction calculation based on the model temperatures.

1.1 Required information

This section outlines the information that is necessary for building an engine thermal model. For this tutorial, the following information is provided, but this section can be used as a reference for future model building.

1.1.1 Geometry

1. Structure (CAD geometry)
    a. Head, block, pistons, crank, head gasket, other significant masses
2. Material properties
3. Water Jacket geometry (optional, can be prepared from structure geometry in GT-SpaceClaim)
4. Surface areas from water jacket geometry used for connections to cylinder structure

1.1.2 CFD - Calibration Data

Input conditions to model

1. Coolant Boundary Conditions
    a. Flow Rate
    b. Inlet Temperature
2. Combustion Boundary Conditions applied to equivalent EngCylStrucCond surfaces
    a. Temperature
    b. Heat Transfer Coefficient
3. Oil Boundary Conditions
    a. Temperature
    b. Heat Transfer Coefficient
    c. Surface area used
4. Ambient Boundary Conditions
    a. Temperature

   b. Heat Transfer Coefficient
   c. Surface area used

Calibration Targets

1. Coolant Pressure Loss vs. Flow Rate
2. Coolant Area Averaged Heat Transfer Coefficient vs. Flow Rate
3. Coolant Heat Transfer Rate vs. Flow Rate
4. Maximum Liner Temperature (for each simulation case)
5. Maximum Dome Temperature (for each simulation case)

1.1.3 Engine model

1. GT-Power Fast running model (with intake / exhaust ports not combined to intake manifold, they are needed for the boundary conditions to the thermal structure)
   a. Part load capable (For example: combustion, injection, spark, valve timings, boost / egr targets included over full operation range)
   b. Transient capable
   c. Necessary controls built-in, capable of running in load mode
   d. Total engine and crank-slider inertias included
   e. Using EngCylTWallSoln
2. Boundary Conditions
   a. Temperatures set to same values in environments and pipe wall boundary conditions

1.1.4 Friction Model

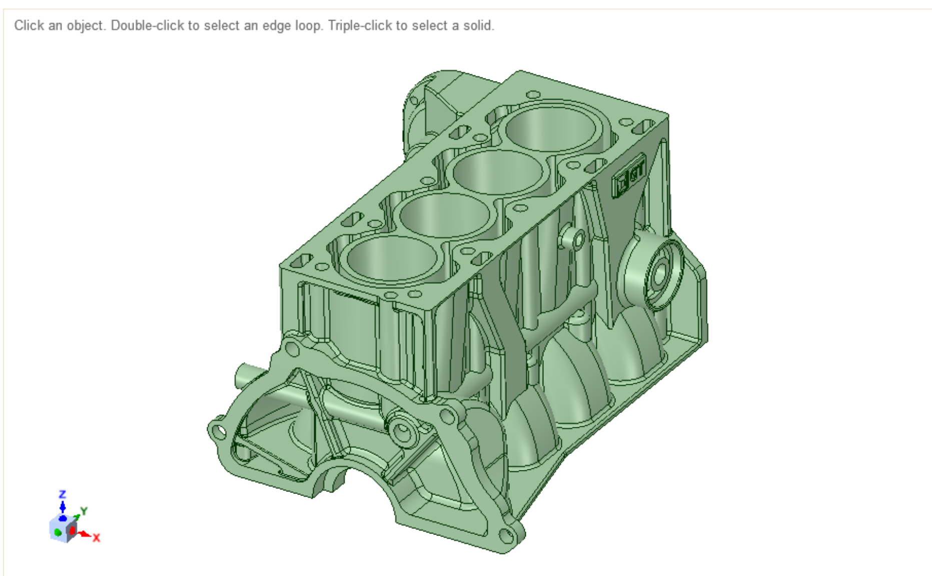
1. Friction values as a function of speed, load, and oil temperature from either:
   a. Strip-down tests
   b. Theoretical Models (Schwarzmeier, Fischer)

1.2 Water Jacket Modeling

The first system we will create is a model of the water jackets from the block and head geometry. We will use GT-SPACECLAIM to extract the water jacket volumes from the material CAD model. It is recommended that you complete the GT-SPACECLAIM tutorials 1 and 2 before beginning this tutorial. Once the water jacket geometry has been created from the original CAD model, GEM3D will be used to characterize the geometry. The solid shapes will be converted to pipe and flowsplit objects, with geometric measurements such as pipe lengths, flowsplit volumes and areas, and expansion diameters obtained from the shapes during the conversion process. These pipe and flowsplit objects in GEM3D will directly translate to parts in GT-ISE, so the 1D model can be built in the 3D environment from the original geometry.

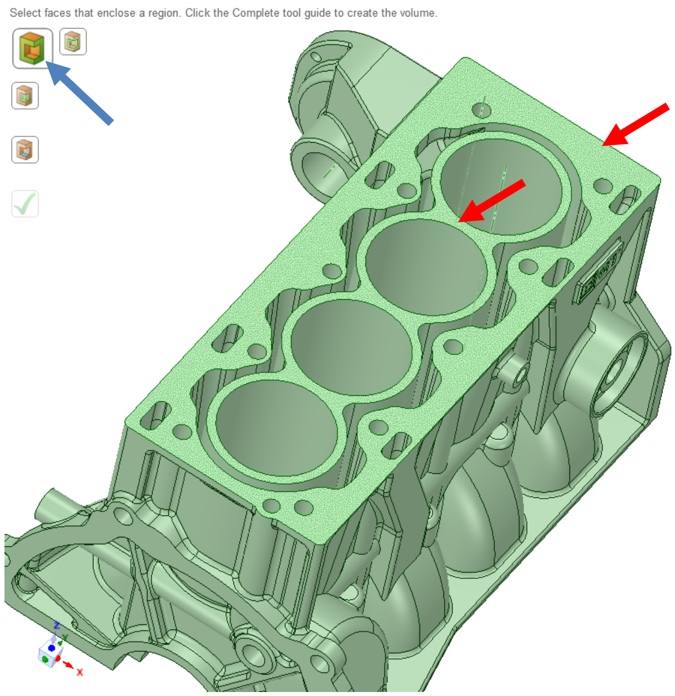
1.2.1 Extracting the Water Jacket Volumes

Open GT-SPACECLAIM from the Tools menu in GT-ISE or GEM3D. Next, open the ACIS file GTI-block-tutorial.sab from the \tutorials\Modeling_Applications\Cooling_Engine\ directory in GT-SPACECLAIM. The file should appear as shown below:

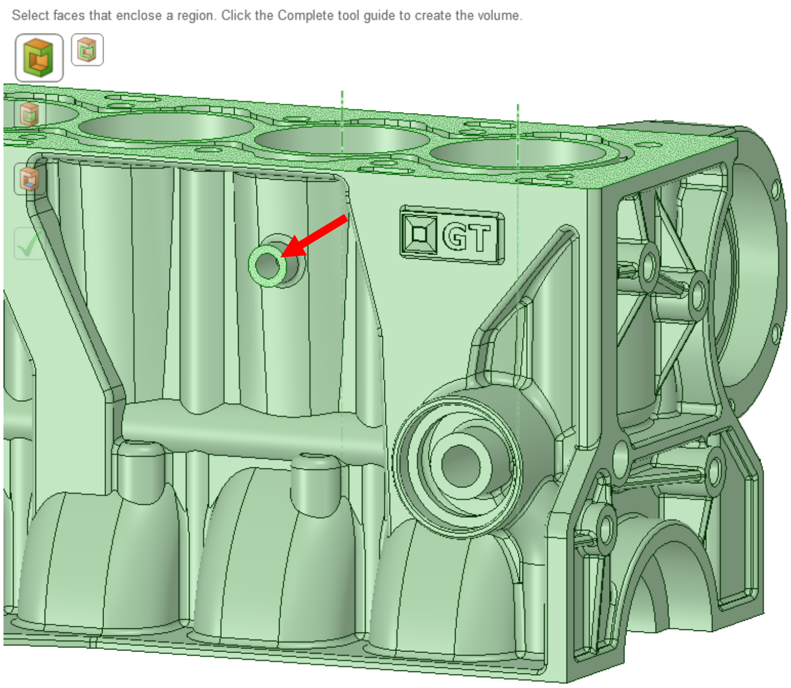


The block water jacket volume can be created using the Volume Extract command, located in the Simplify menu. Click on this button to begin the operation.

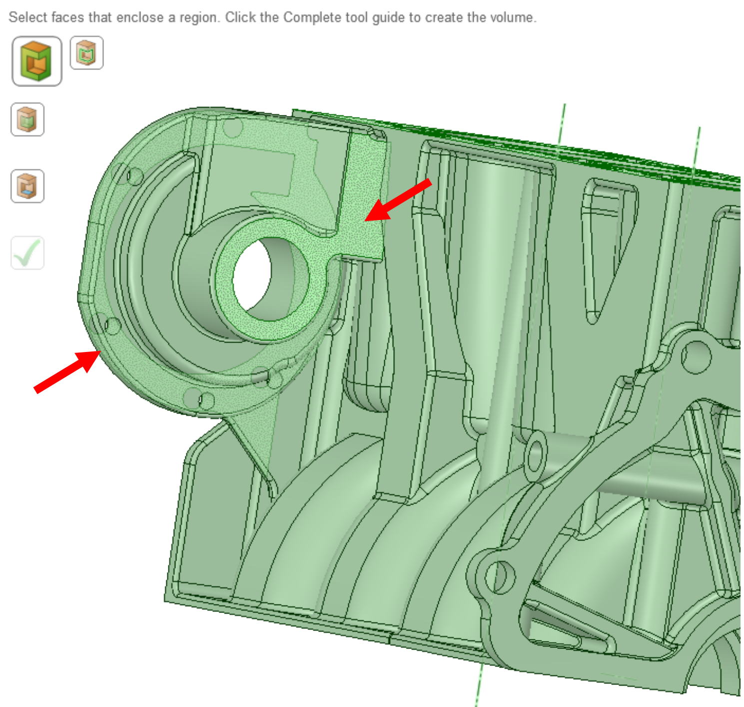
The first step in the Volume Extract operation is to select the boundary surfaces which enclose the water jacket volume. In this model, there are five boundary faces which enclose the block water jackets. Surfaces 1 and 2 are on the top of the block, left-click to select them.



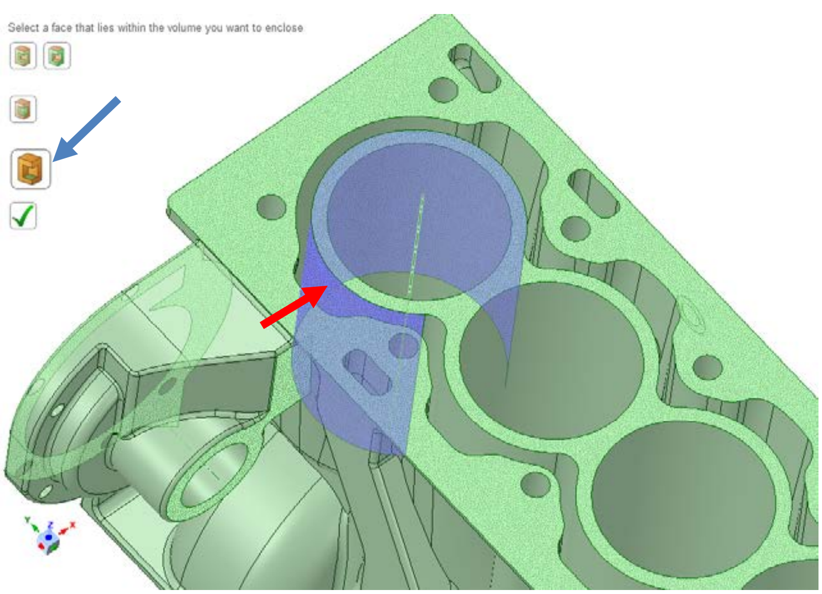
Surface 3 is on the +X side of the block, surrounding the outlet for the EGR cooler branch.



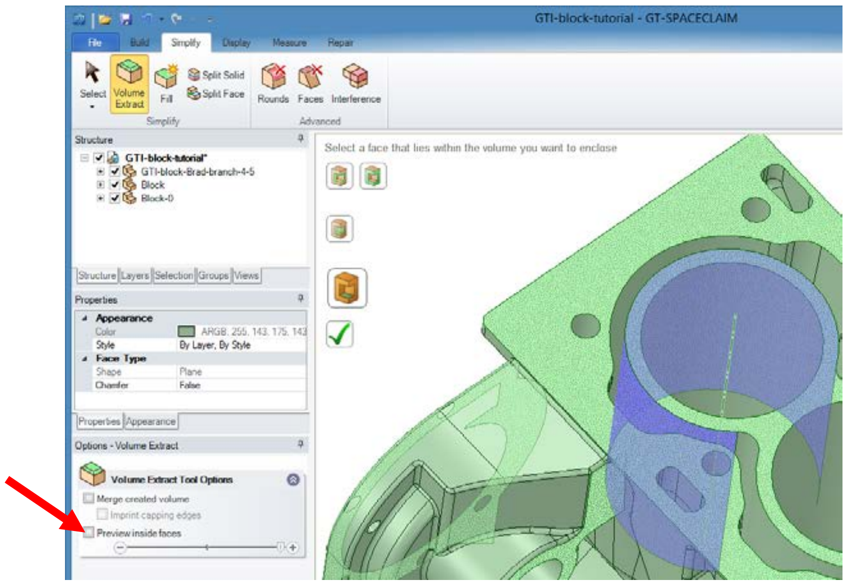
Surfaces 4 and 5 are on either side of the water pump housing, on the -X side of the block.



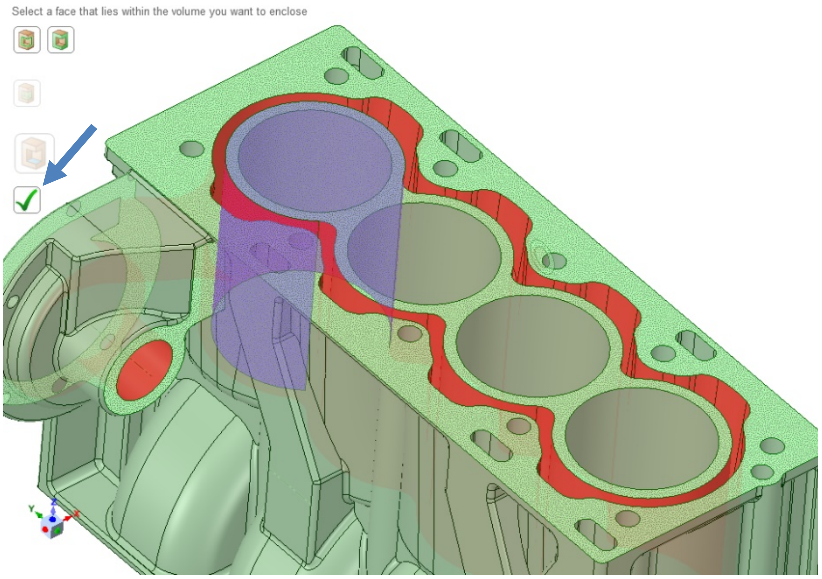
When the boundary surfaces have been selected, switch to the seed face tool guide by clicking on the icon noted by the blue arrow below. Select one of the internal surfaces of the water jacket to be the seed face, which should show up in blue:



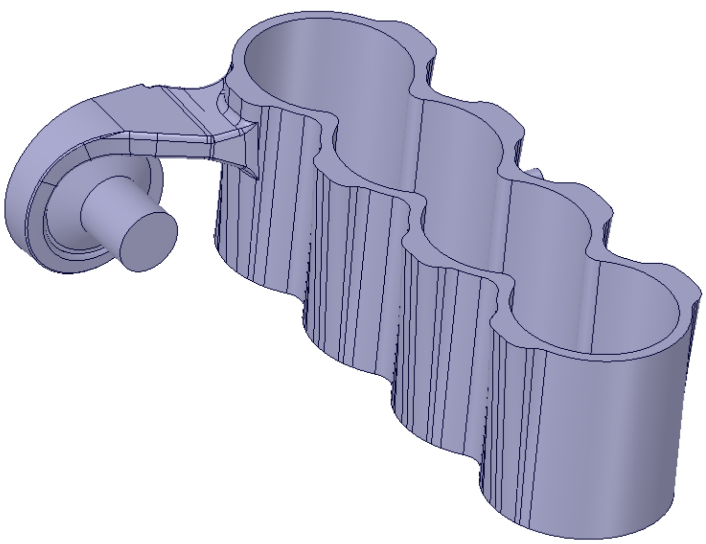
After the seed face is selected, preview the inside faces of the water jackets to ensure that the correct boundary surfaces are selected. This option is available in the left hand toolbar, as shown in the diagram below.



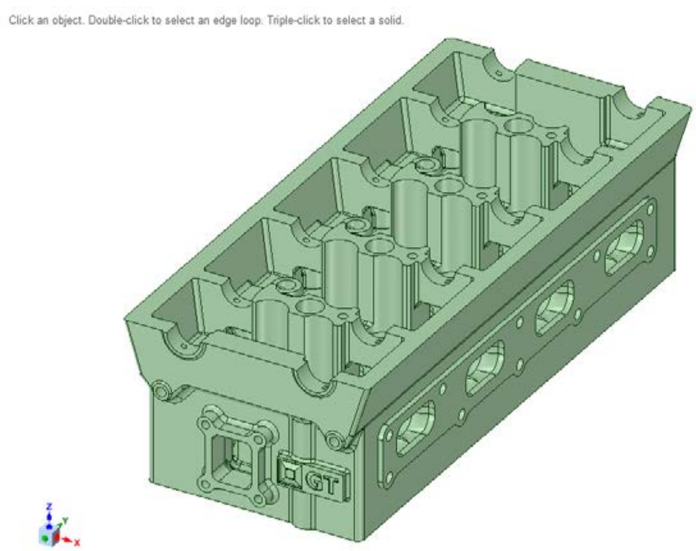
The red surfaces represent the surfaces that will be used to create the new water jacket volume. If the red surfaces are completely contained and match up to the actual water jackets, select the green check mark as indicated by the blue arrow in the figure below to complete the Volume Extract operation and create the new volume.



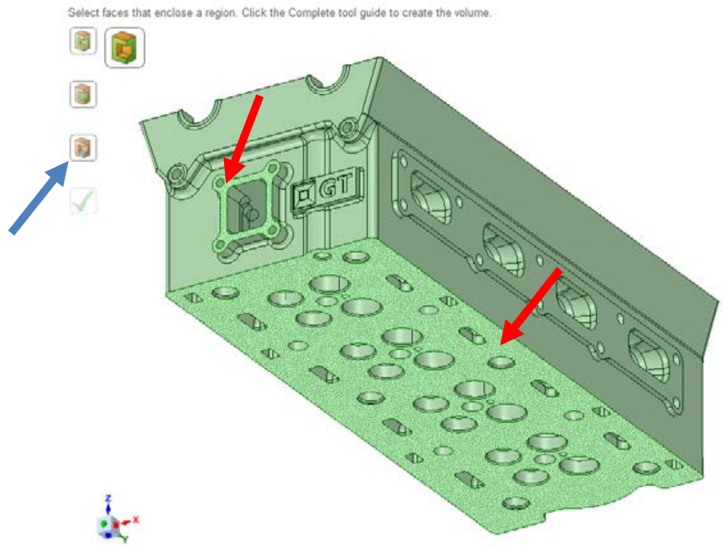
Once the water jackets are complete, turn off the block solid body "Solid1" by unchecking the box beside the structure name in the upper left corner of the GT-SpaceClaim window and save the Volume as an ACIS file (Block-WJ.sab). Any geometry turned off in the GT-SpaceClaim file will be excluded from the ACIS file. We will import this new file into GEM3D after we prepare the head water jackets.



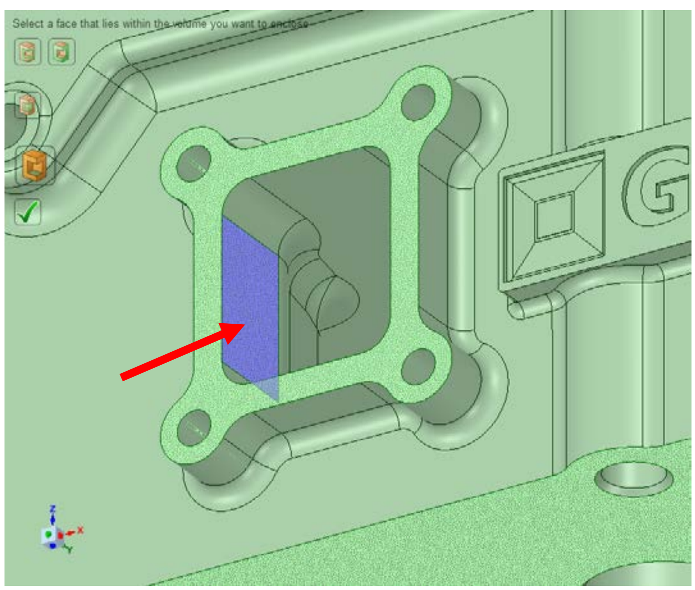
The next step is to extract the water jacket volumes from the cylinder head. Open up the file named GTI-cylhead-tutorial.sab located in \tutorials\Modeling_Applications\Cooling_Engine\ directory using GT-SPACECLAIM. The file should appear as shown below:



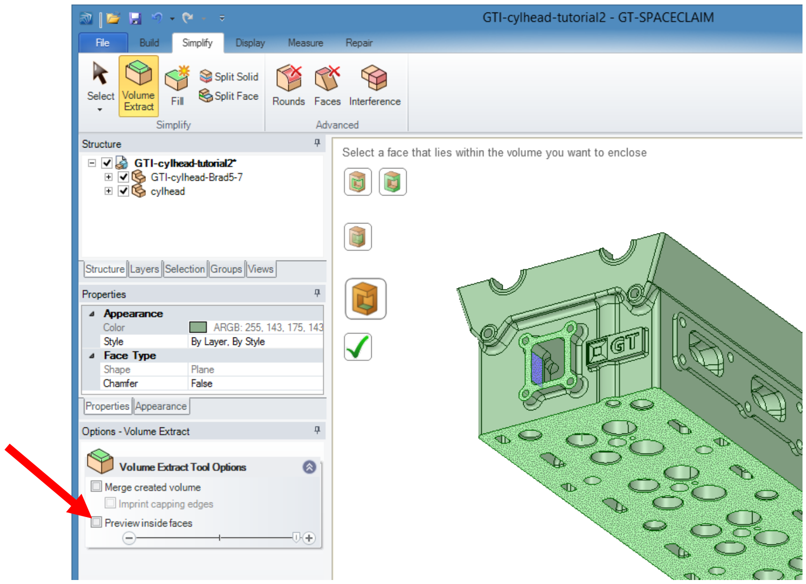
Next, begin the Volume Extract operation from the button in the Simplify menu. Two boundary surfaces are needed to fully enclose the water jackets in the head, they are shown on the image below. After the boundary faces are selected, switch to selecting the seed surface using the tool guide marked with the blue arrow in the image below:



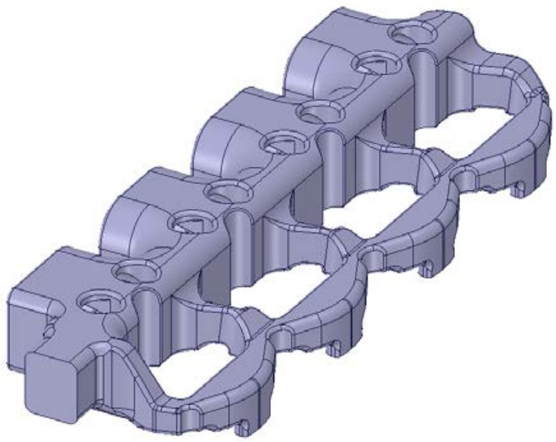
Select one of the internal surfaces of the water jackets to be the seed surface for the volume, it should appear in blue:



After the seed face is selected, preview the inside faces of the water jackets to ensure that the correct boundary surfaces are selected. This option is available in the left hand toolbar, as shown in the diagram below.

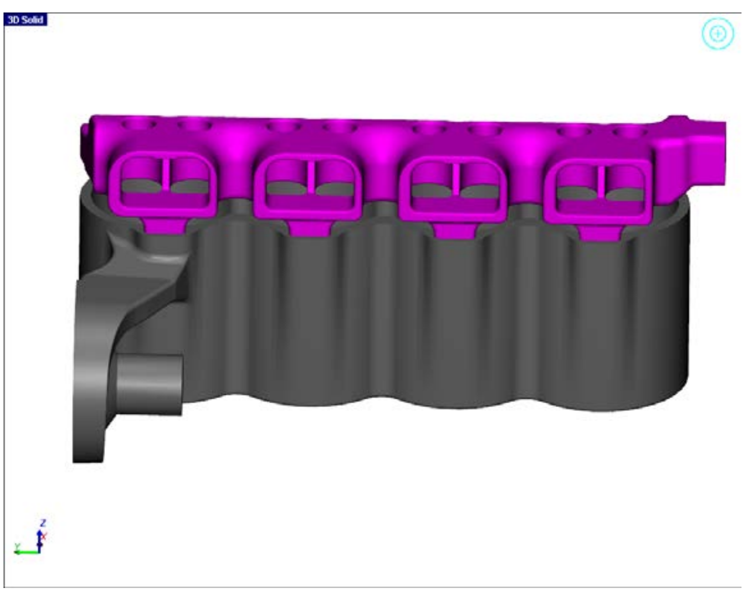


The red surfaces represent the surfaces that will be used to create the new water jacket volume. If the red surfaces are completely contained and match up to the actual water jackets, select the green check mark to complete the Volume Extract operation and create the new volume. Once the water jackets are complete, turn off the cylinder head solid body "Solid1" by unchecking the box beside the structure name in the upper left corner of the GT-SpaceClaim window and save the Volume as a ACIS file (Head-WJ.sab).



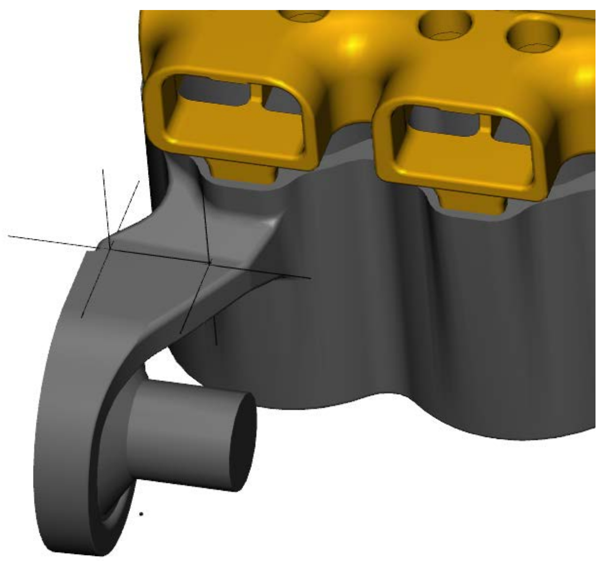
1.2.2 Converting the Water Jackets in GEM3D

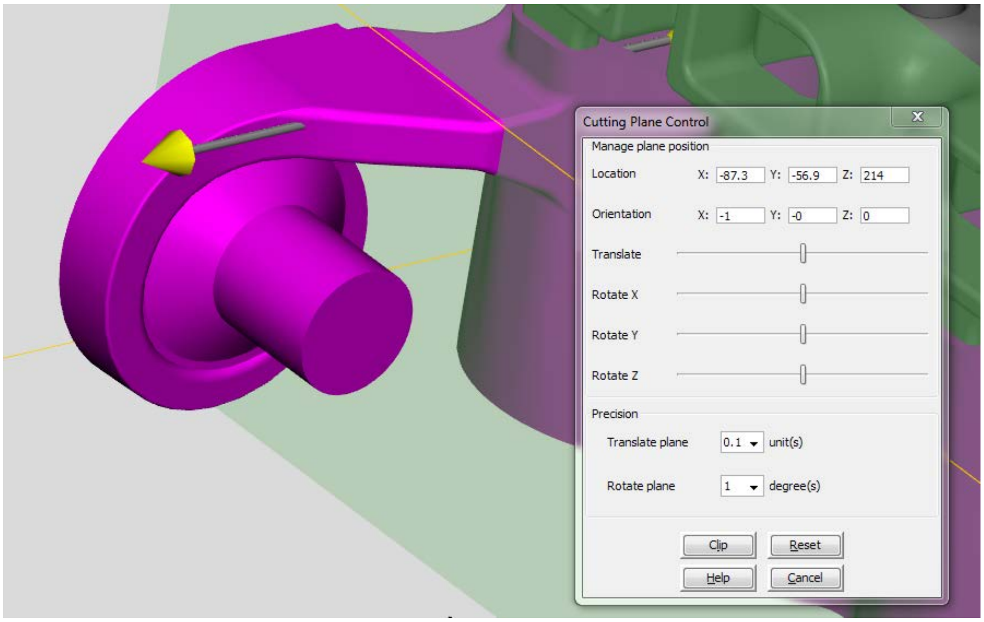
The next step in the model building process is to convert the extracted volumes into flow parts. To begin, either open GEM3D from the Tools > GT Applications menu in GT-ISE and use the File > Import 3D… command, or use the 1-click import button from GT-SPACECLAIM (in the "Build" menu) to automatically import the currently displayed geometry to GEM3D. Import both the head and block water jacket files, the model should appear as shown. Save the model as water-jackets-tutorial.gem.

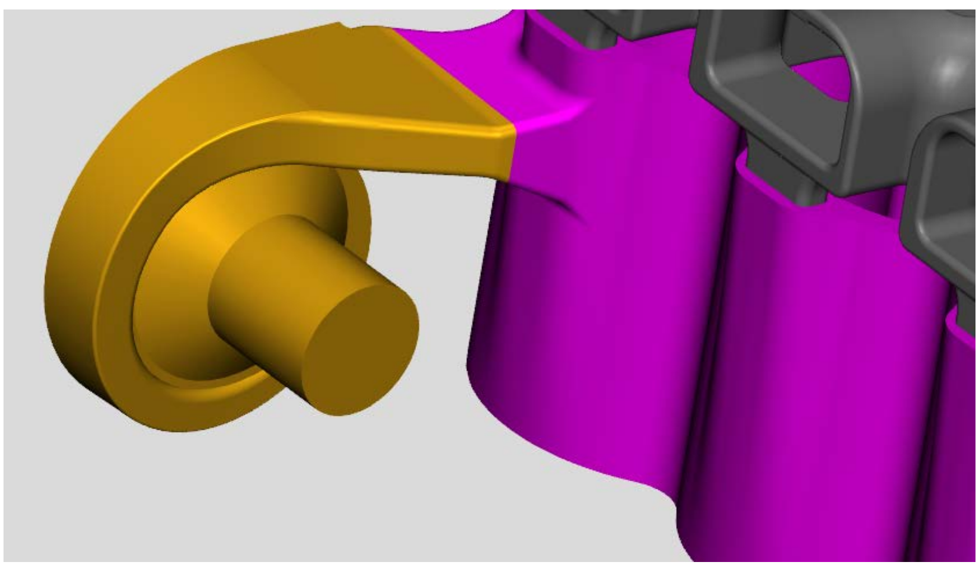


Before converting the imported solids to flow parts, they need to be divided into pipes and water jacket volumes. The pipes that are separated will be automatically connected to the water jacket volumes when the model is exported from GEM3D, and will serve as the inlet and outlet connections to the water jacket volumes. The pipes will be connected to either boundary condition parts or the rest of the coolant circuit later on in the tutorial.

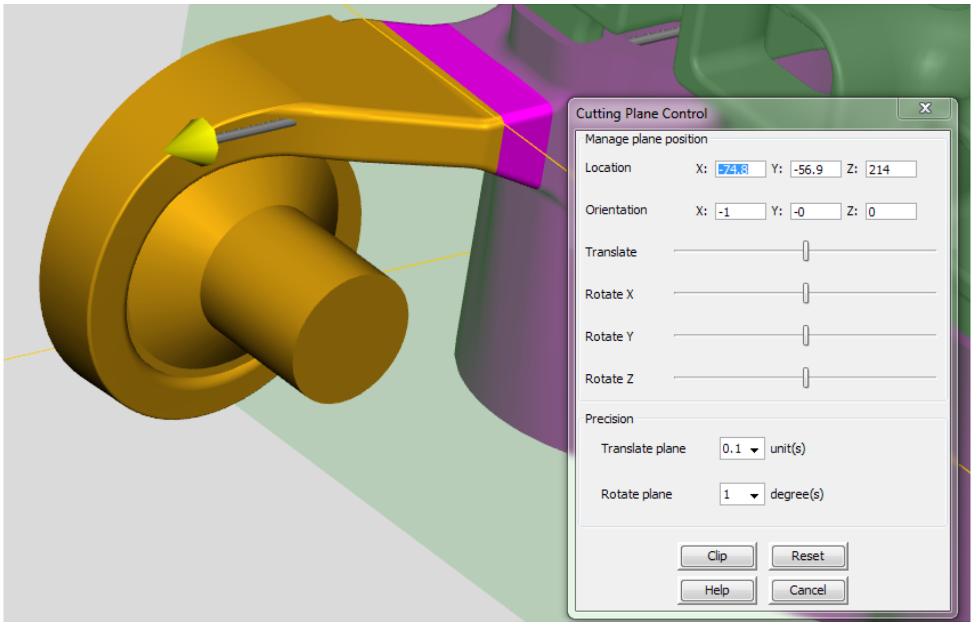
Cutting tools are available to divide up the solid geometry. First, separate the pump inlet part from the water jackets using the Cutting Plane tool, selected from the toolbar.  Place 3 points on the block water jacket solid to create a cutting plane, select the geometry to be cut, and then click the Clip button to split the geometry.

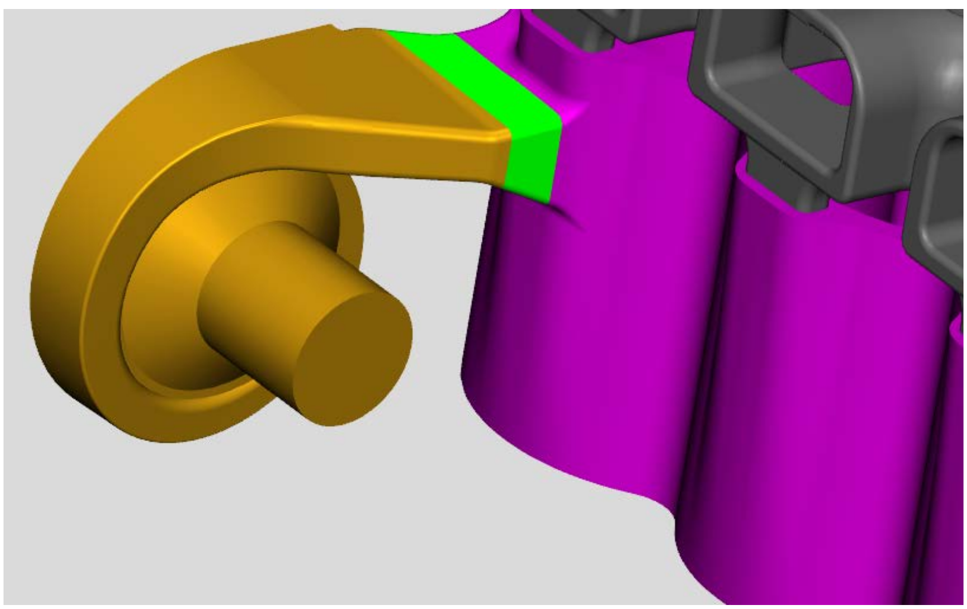




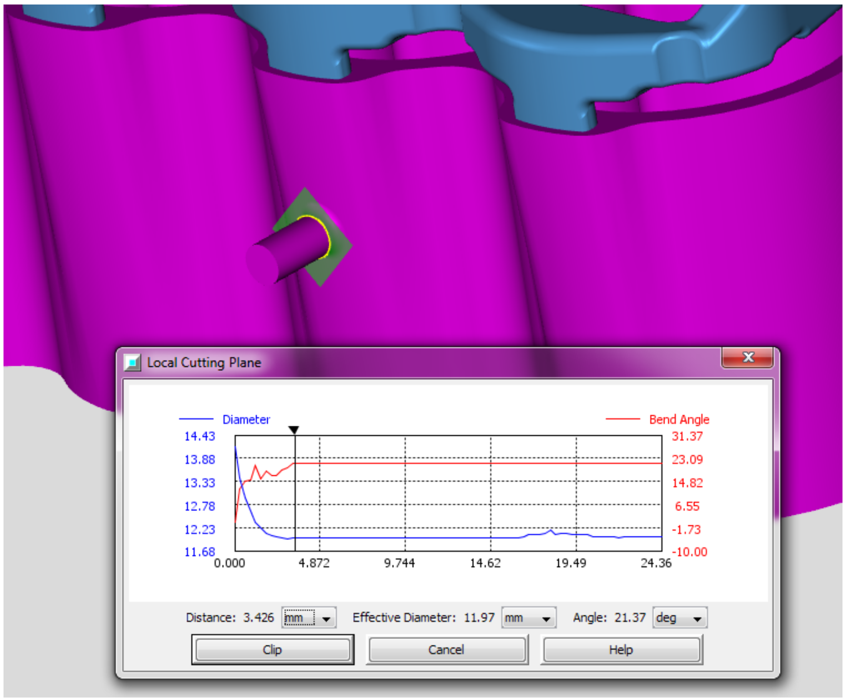


Next, separate another section of the water jacket inlet part. The previously used cutting plane can be returned using the Restore Cutting Plane function.  Restore the previous cutting plane, and drag/move it closer to the water jackets. Select the block water jacket solid, and click the Clip button to separate a section of the inlet part as shown:

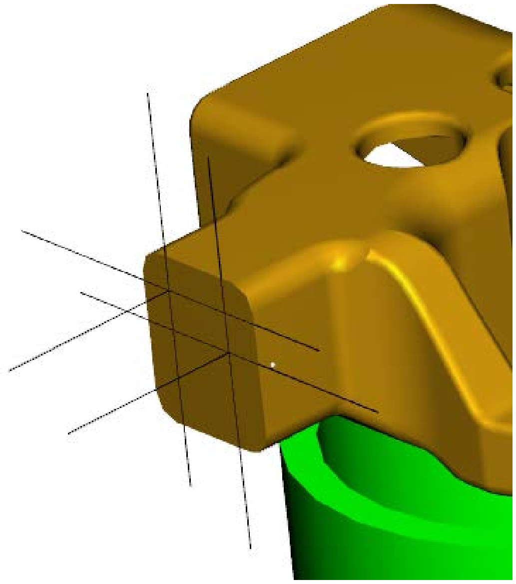


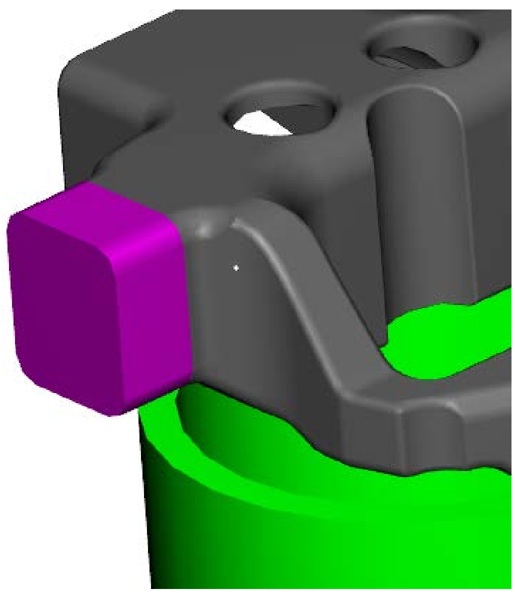


The next pipe to slice is the outlet which goes to the EGR cooler. To separate this section, use the Local Cutting Plane tool. This tool analyzes the center line of a shape, and follows the center line so that cuts perpendicular to the flow direction can be made. A diagram showing the bend angle and diameter along the pipe is given so that appropriate cuts can be made. To begin, select the Local Cutting Plane from the toolbar and then select a point on the EGR outlet pipe as shown. Drag the cutting plane along the length of the pipe by either dragging the plane directly or dragging the arrow indicator in the "Local Cutting Plane" dialog, and click Clip to separate the solid.

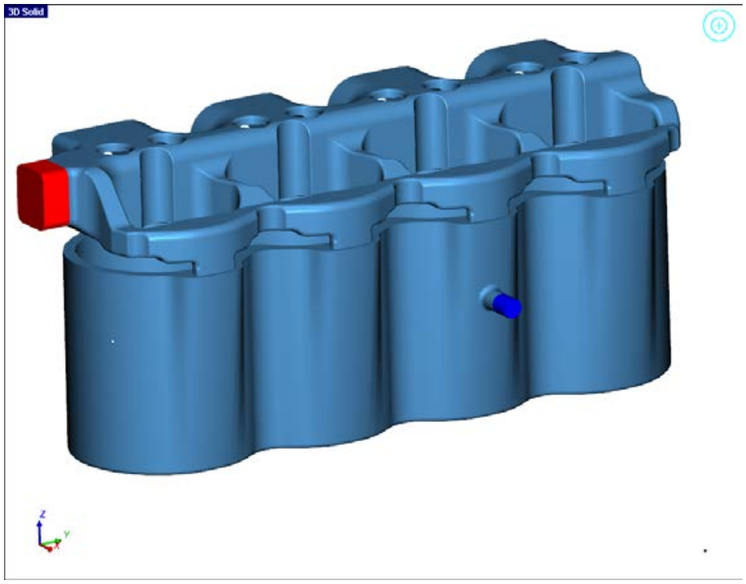


The last separate shape to make is at the outlet of the head. Create a new cutting plane by placing three points on the planar surface at the head outlet, as shown below. Then, move the cutting plane to slice a small section of the solid, and click the Clip button to separate the solids.

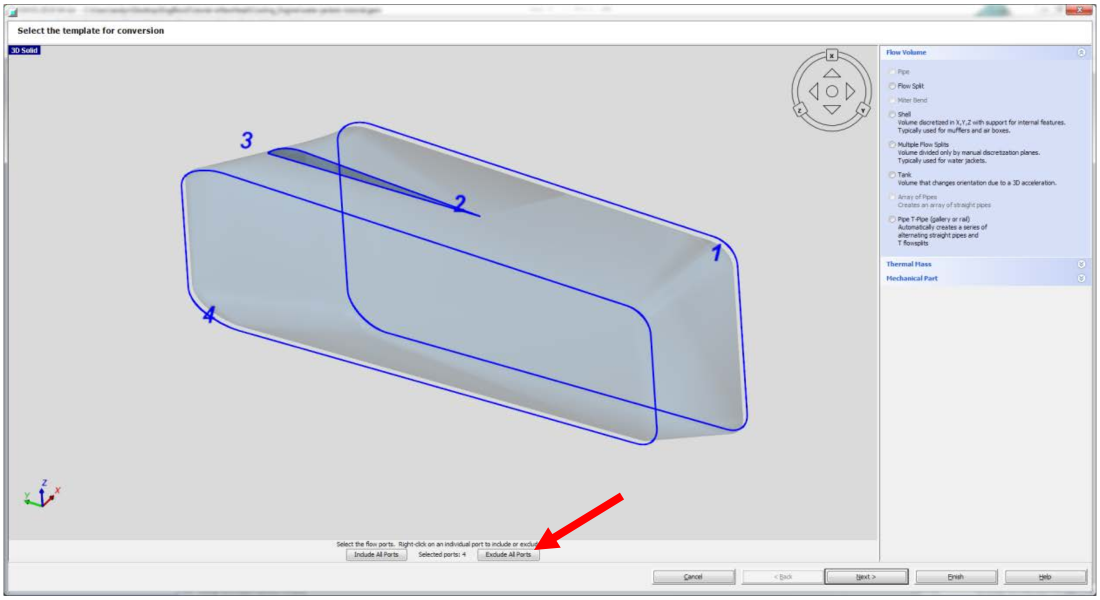




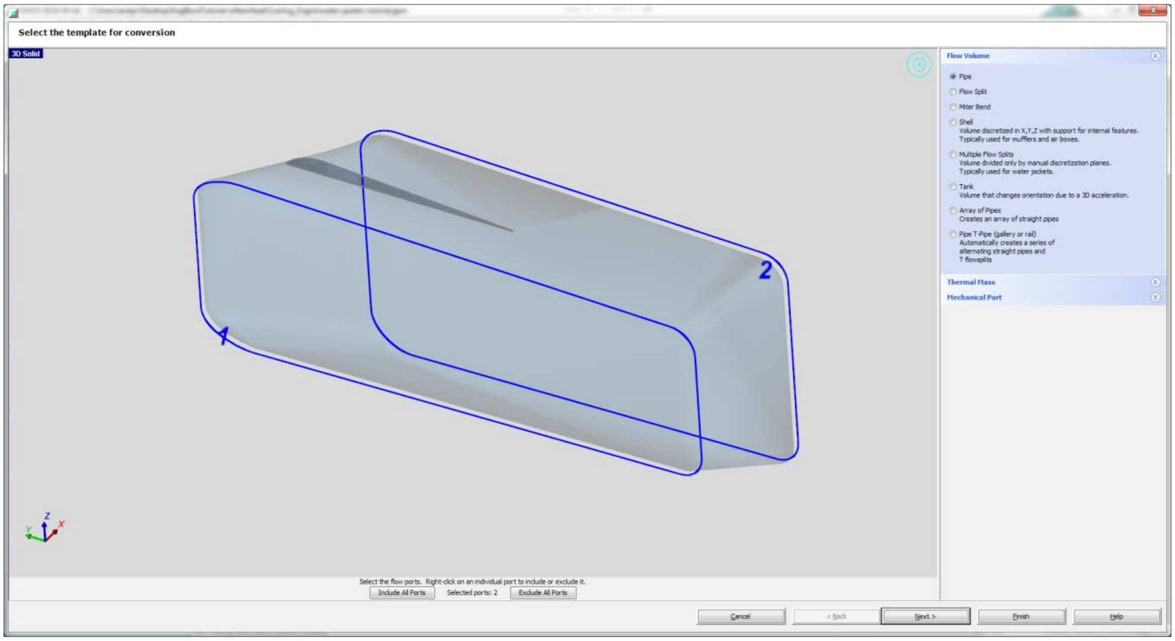
The last step in preparing the geometry for conversion is to merge the block and head water jackets, excluding the pipes which were just separated, into a single solid shape. This will allow the connections between the block and the head water jackets to be automatically created when the model is exported. To combine the two solids, select them both while holding the CTRL key, and then click the Merge button in the toolbar,  or use the Slicer--Merge Meshes command. Your model should appear as shown, although the colors may be slightly different.



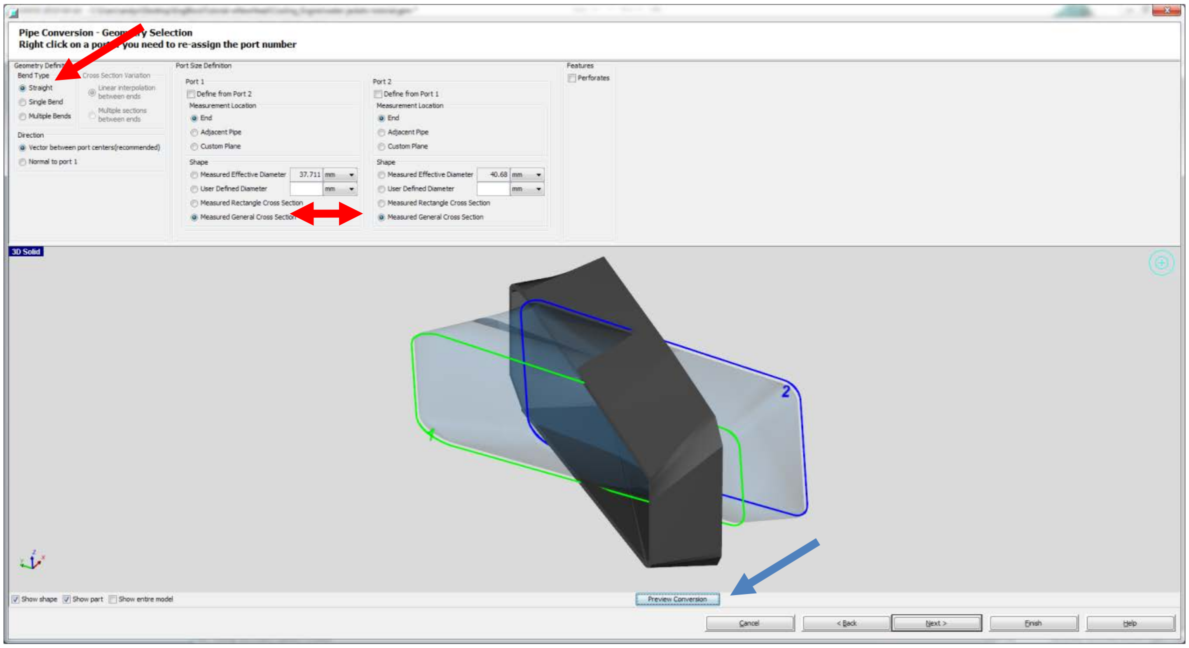
Now that the model has been separated, the next step is to convert the Solid Shapes to GEM3D Components, so that they will be modeled in GT-ISE as pipes and flowsplits. This command is available either by selecting a part and right-clicking on it, or by choosing the Convert Shape to Component command in the toolbar.  The first shape to convert is the part at the inlet to the water jackets. Select it, and open the conversion wizard through either the Convert Shape toolbar button or command in the right-click menu. The wizard should appear as shown below:



The conversion wizard analyzes the selected geometry, and highlights the potential ports. For mesh shapes, this includes all gaps or openings in the mesh surface. For solid shapes, all planar surfaces are highlighted. For this shape, several surfaces are selected, but only 2 are needed for a pipe conversion. Use the Exclude All Ports button at the bottom of the 3D window to clear all currently selected ports. Then, right-click on the two ports at either end of the pipe and select "Include Port" to add them to the selection. Your window should appear as shown:



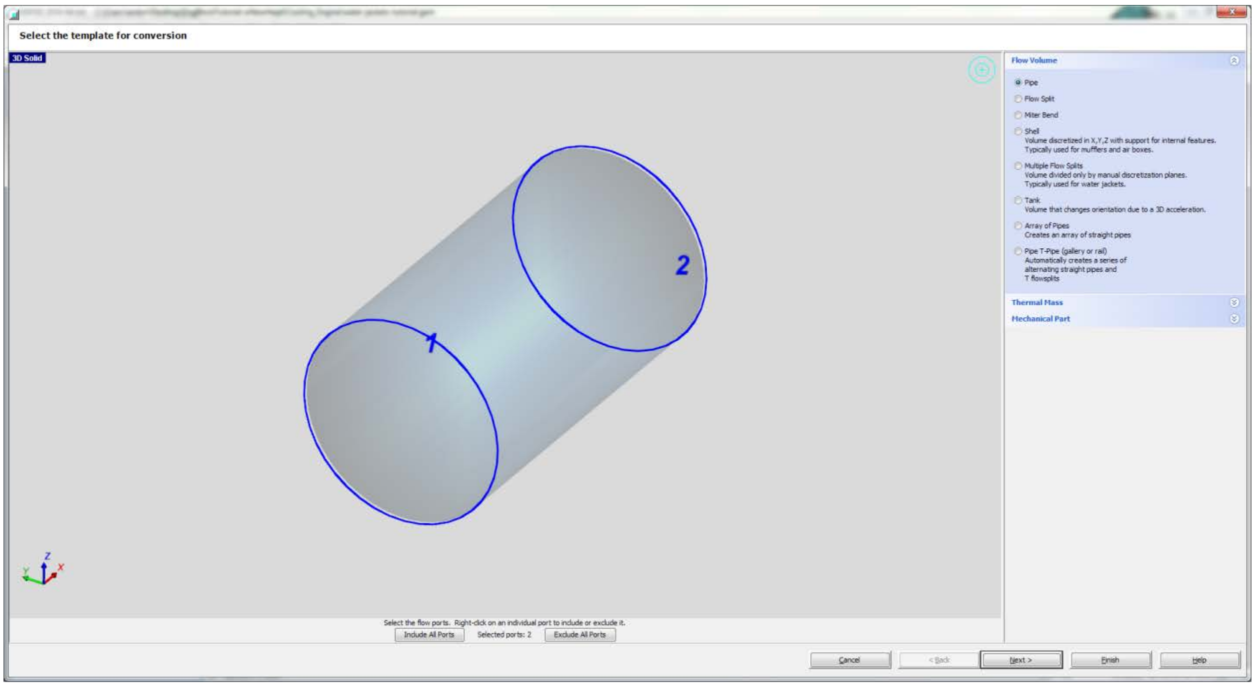
Select the "Pipe" option from the"Flow Volume" conversion options on the right, and click Next to continue to the pipe creation screen. In the Geometry Definition section, select a Straight Pipe. In the Port Size Definition section, select the Measured General Cross Section for both Port 1 and Port 2. Click the Preview Conversion button, and the screen should appear with the proposed conversion.



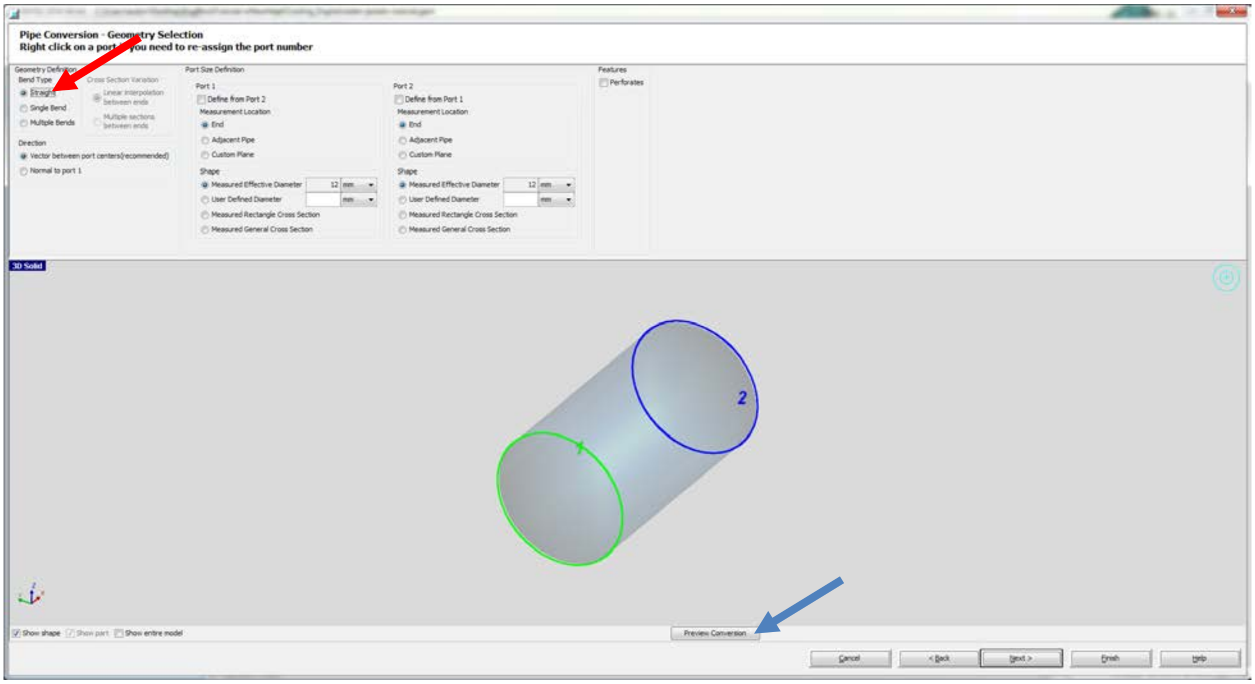
Note that for this case, the pipe preview does not match the port direction of the original geometry. The direction is instead determined by a vector drawn between the centers of the two ports. This vector is used to determine the pipe length, and the inlet and outlet port planes are normalized to this vector. The important values to confirm for the conversion are that the pipe center line follows the original geometry, and that the inlet and outlet port cross sections match the original geometry.

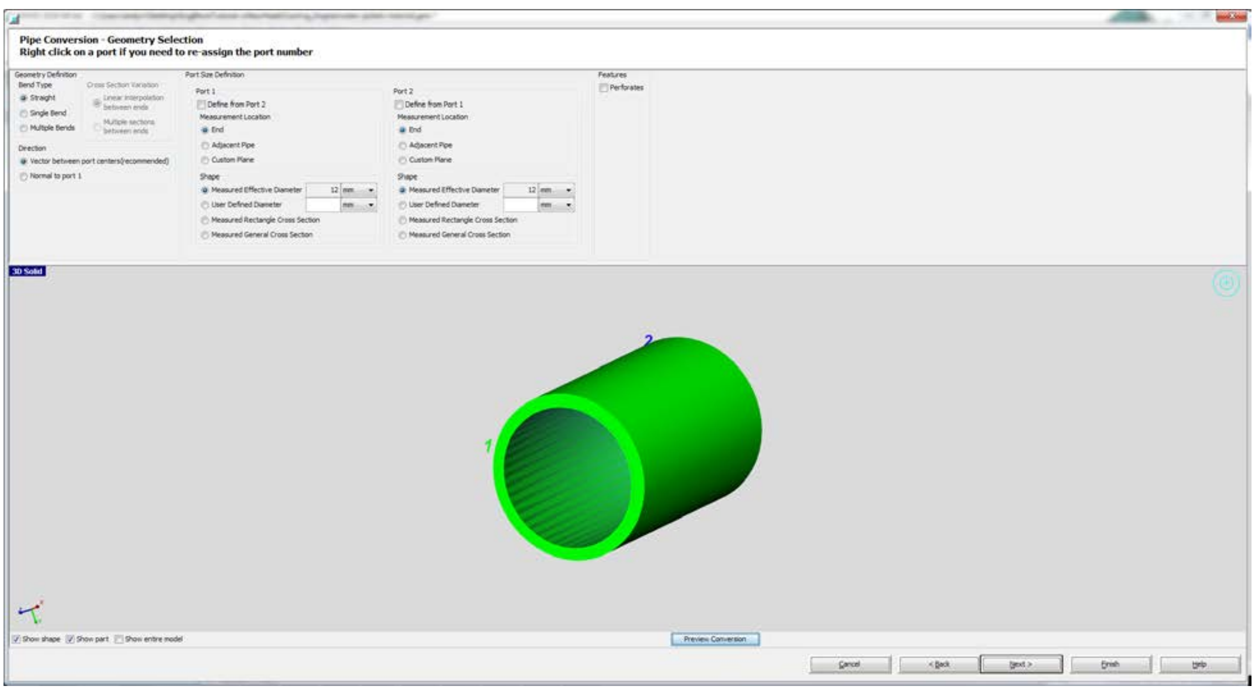
If the preview geometry appears correctly, click the Next button to continue. The 3rd screen in the wizard is the" Calculated Geometry" screen, which displays the values that are calculated from the original geometry. The Next button will bring up the final screen, where the part name can be defined along with the material properties, the thermal modeling options, and pressure drop modeling options. Name the pipe "Inlet-Pipe" and set the **Initial State Name** to "Coolant-Initial" in the Main folder. This will create a reference object for the fluid used in this pipe, which we will define before running the simulation later in the tutorial. In addition, select the **Wall Temperature from Connected Thermal Primitive** option in the Thermal folder of the pipe. Click the Finish button to complete the conversion.

The next two pipes to convert are the EGR outlet pipe and the Head outlet pipe. Right-click on the solid shape for the EGR outlet pipe, and open the conversion wizard using the Convert Shape to Component command.



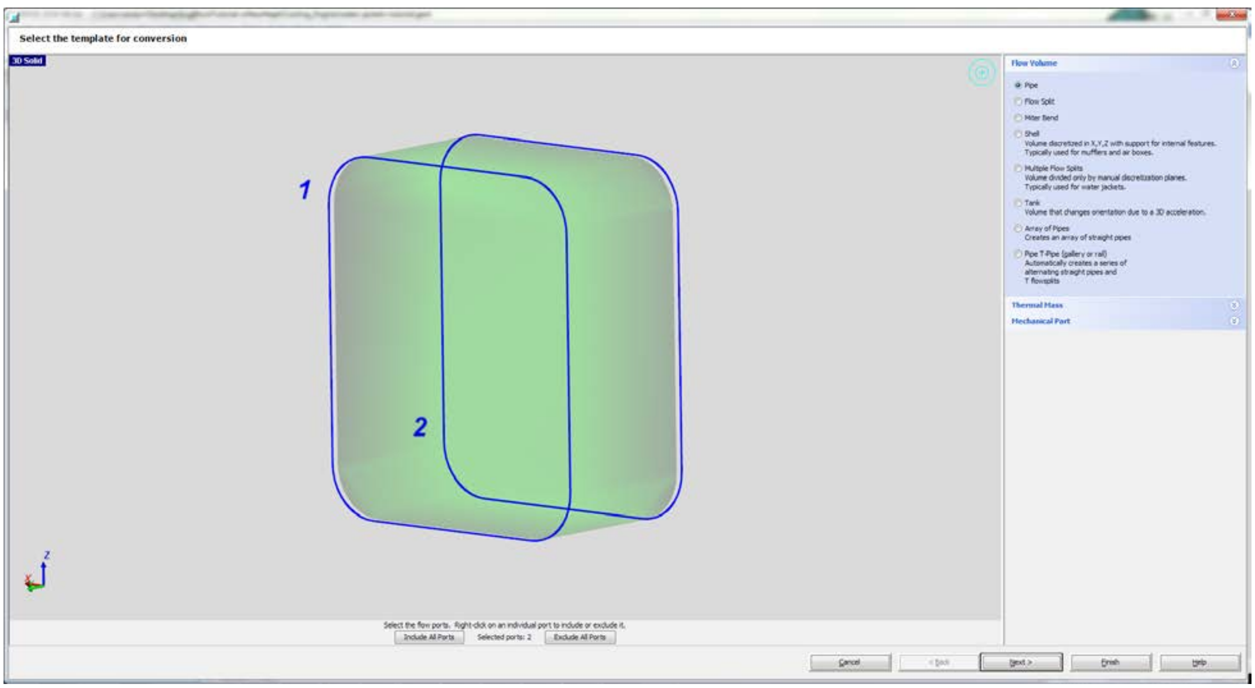
Only 2 ports are detected for this geometry, so select the Pipe option and click Next. Select the Straight option for the Bend Type, and then click the Preview Conversion button. If the created geometry matches well to the original solid shape, click the Next button to continue.


Pipe Conversion - Geometry Selection
Right click on a port if you need to re-assign the port number
Geometry Definition
Bend Type
Straight
Single Bend
Multiple Bends
Direction
Vector between port centers(recommended)
Normal to port 1
Port Size Definition
Port 1
Define from Port 2
Measurement Location
End
Adjacent Pipe
Custom Plane
Shape
Measured Effective Diameter
User Defined Diameter
Measured Rectangle Cross Section
Measured General Cross Section
Port 2
Define from Port 1
Features
Perforates
3D Solid
Show shape
Show part
Show entire model
Preview Conversion
Cancel
< Back
Next >
Finish
Help

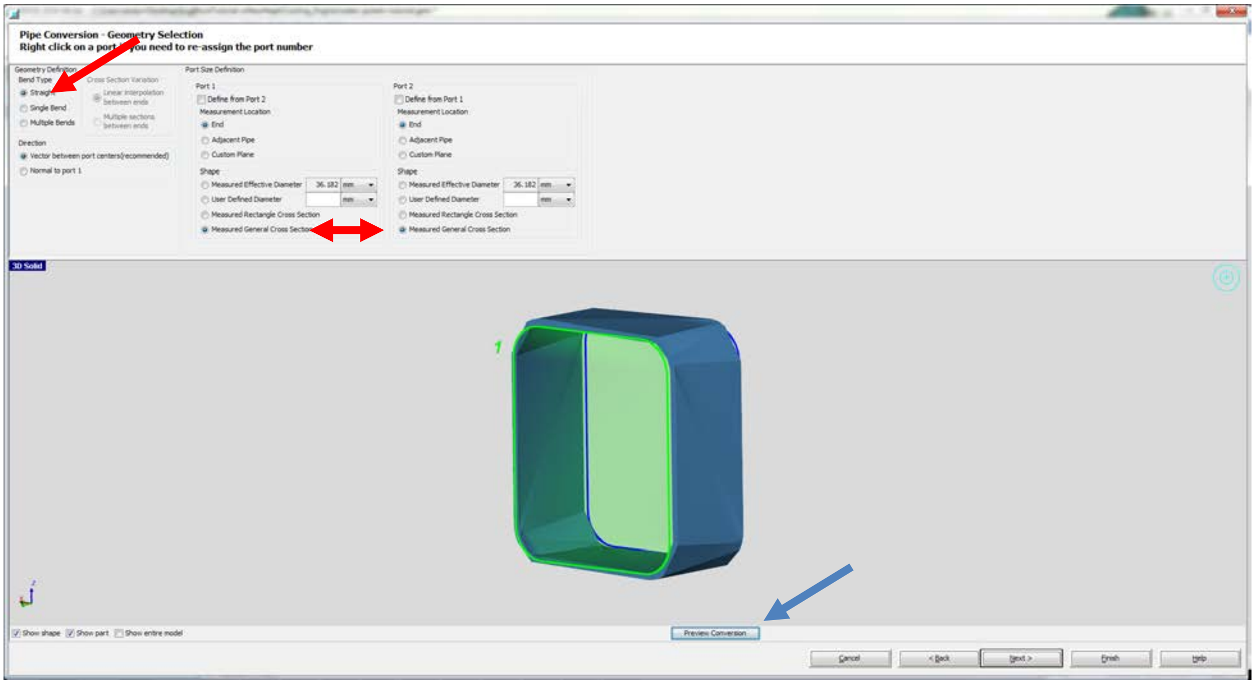

Pipe Conversion - Geometry Selection
Right click on a port if you need to re-assign the port number
3D Solid
Preview Conversion
Cancel
< Back
Next >
Finish
Help

Review the calculated geometry to make sure the shape matches what is displayed on the screen, and click Next to continue to the final screen. The previously selected options for the **Initial State Name** and Wall Temperature Method.should be selected in the new pipe. If not, change them so the **Initial State Name** is set to "Coolant-Initial" and the Wall Temperature Method is set to **Wall Temperature from Connected Thermal Primitive**. Name the pipe EGR-Outlet, and click the Finish Button to complete the conversion.

For the head outlet pipe, select it and right-click to open the conversion wizard. Extra ports may be detected due to the additional planar surfaces; if so, first use the Exclude All Ports button to clear the selection, and include the 2 ports on either end as shown below. Select the Pipe option from the right hand side, and click Next to continue to define the pipe.



In the Geometry Definition section, select a Straight Pipe. In the Port Size Definition section, select the Measured General Cross Section for both Port 1 and Port 2. Click the Preview Conversion button, and the screen should appear with the converted geometry.

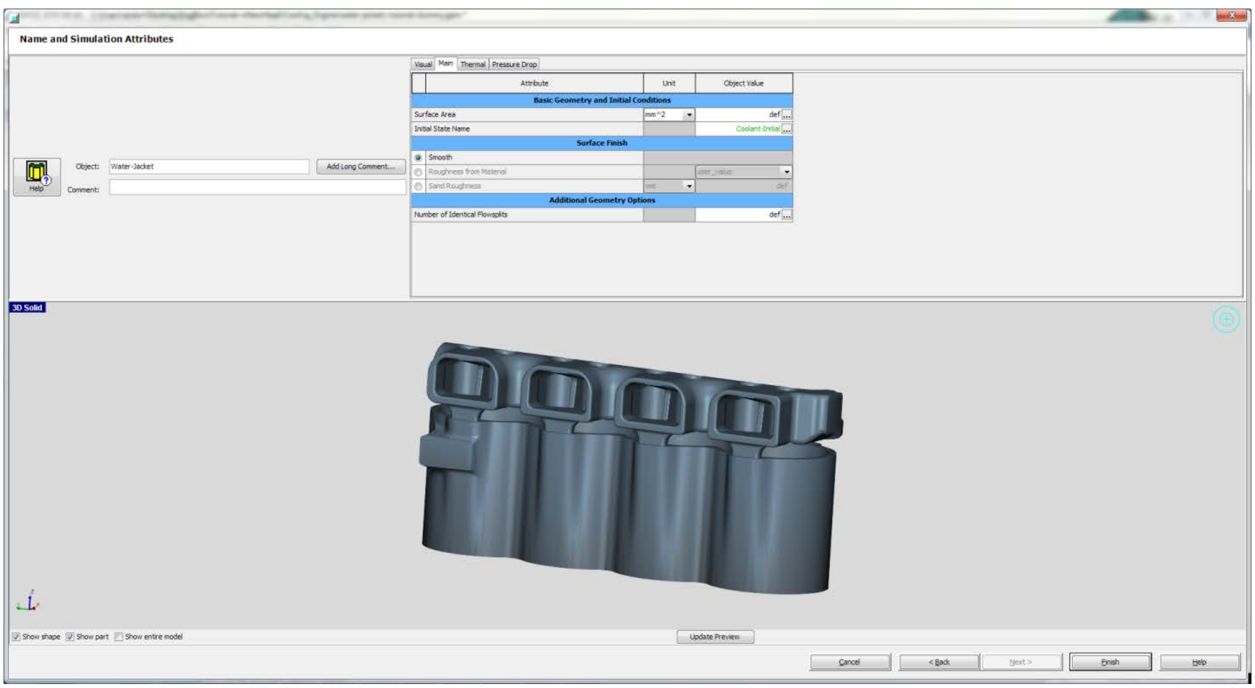


If the preview geometry appears correctly, click the Next button to continue. Review the calculated geometry to make sure the shape matches what is displayed on the screen, and click Next to continue to the final screen. The previously selected options for the **Initial State Name** and Wall Temperature Method.should be selected in the new pipe. If not, change them so the **Initial State Name** is set to "Coolant-Initial" and the Wall Temperature Method is set to **Wall Temperature from Connected Thermal Primitive**. Name the pipe Head-Outlet, and click the Finish Button to complete the conversion.

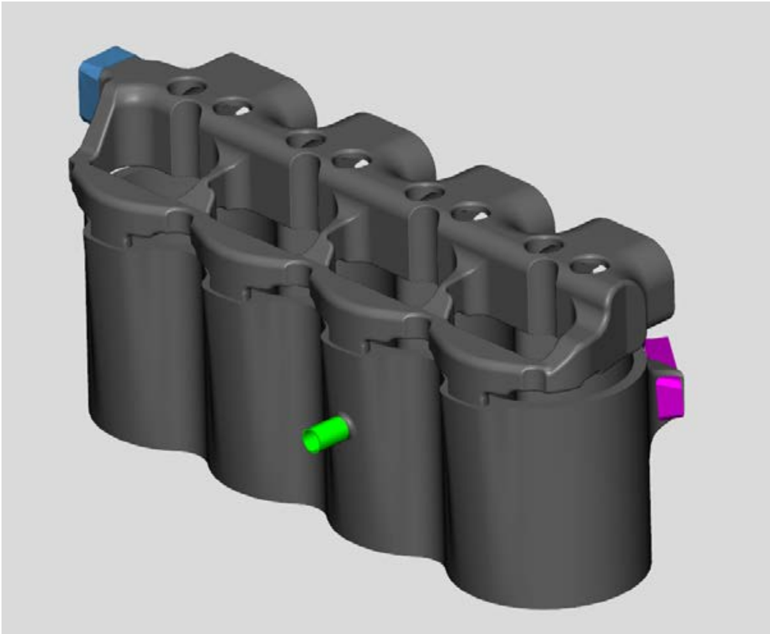
The remaining geometry to be converted is the main water jacket volume. Rather than slicing the geometry up into much smaller pipes and flowsplits segments, the geometry will be converted to a single GEMSolidFlowVolume, which represents a general volume that can be divided into multiple smaller flowsplits and connected to adjacent pipes. The water jackets will be divided into segments using datum planes, so that a single cylinder will connect to one head flow volume and one (cyls 1 & 4) or two (cyls 2 & 3) block flow volumes. This method gives a reasonable value for the average temperature distribution adjacent to a given cylinder, but allows for fast model creation and calibration. Further sub-dividing the model is possible depending on the level of interest.

To begin the conversion, right-click on the water jacket volumes and select the Convert Shape to Component command. Many ports will be detected since there are a large number of planar surfaces on this solid, but in this case, no ports are necessary; they will be automatically created between the volumes when the datum planes are added, and where the already-converted pipes conect to the volume. Use the Exclude All Ports button at the bottom of the 3D window to clear the currently selected ports, and select the Multiple Flow Splits option from the right hand side before clicking Next.

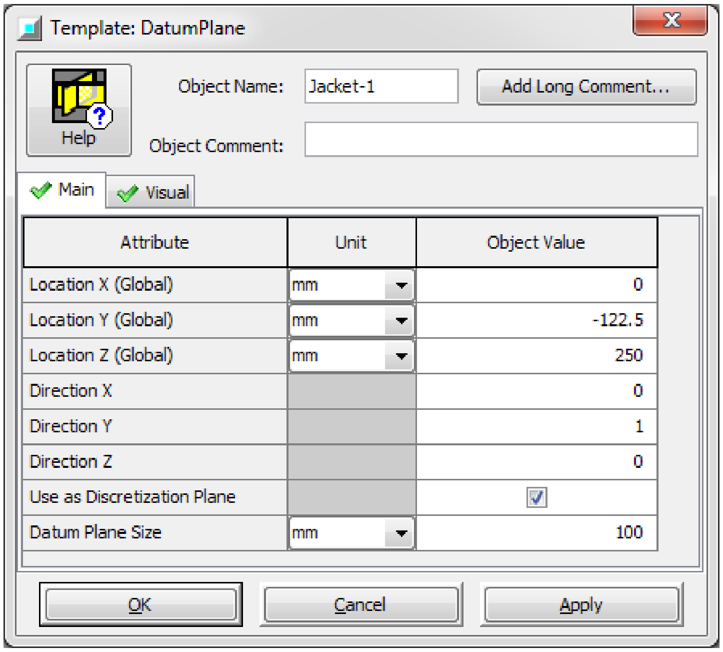
The **Name and Simulation Attributes** screen should appear, with the converted volume appearing in the 3D window. Enter Water-Jacket for the object name, and check the Main folder to ensure that Coolant-Initial is used for the the **Initial State Name**. Also, check the Thermal folder to make sure that the **Wall Temperature from Connected Thermal Primitive** option is selected. If those options are correct and the name is entered, click the Finish button to create the volume.



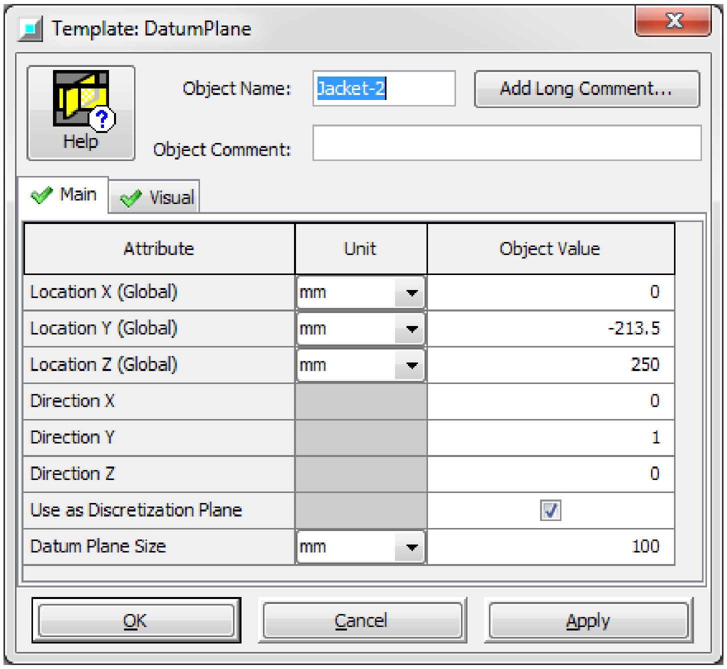
Turn off any remaining solid shapes in the model, and the converted geometry should appear as shown:

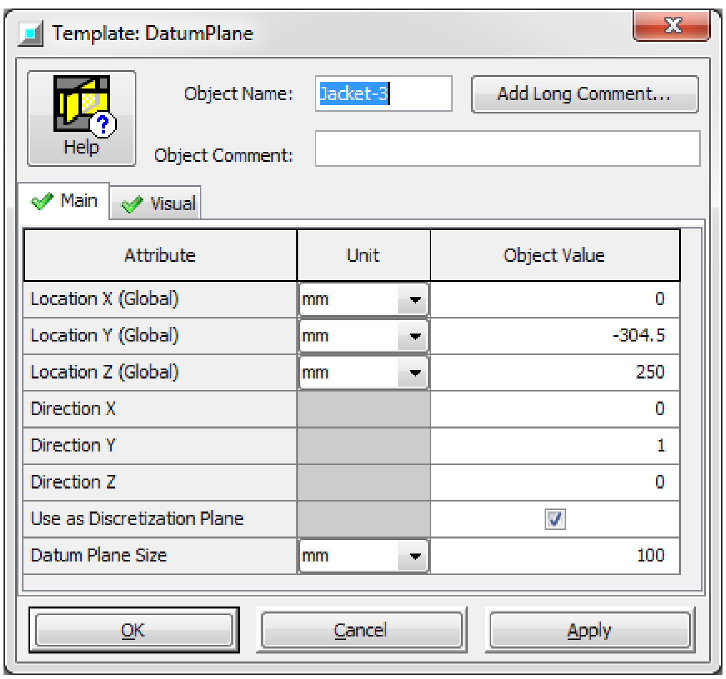


The next step is to add Datum planes to the GEMSolidFlowVol, to divide the single volume into multiple volumes as discussed previously. Select the volume, and add a datum plane from the Builder > Add Datum Plane > Child Datum Plane command. This will create a new datum plane which will divide the shell based on the location and direction entered for the plane. Name the object Jacket-1, and enter in the datum plane values as shown below:

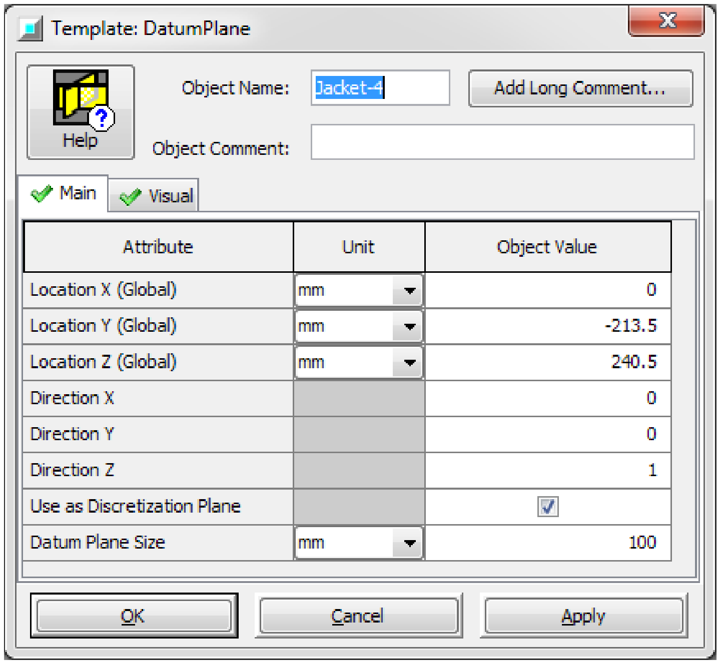


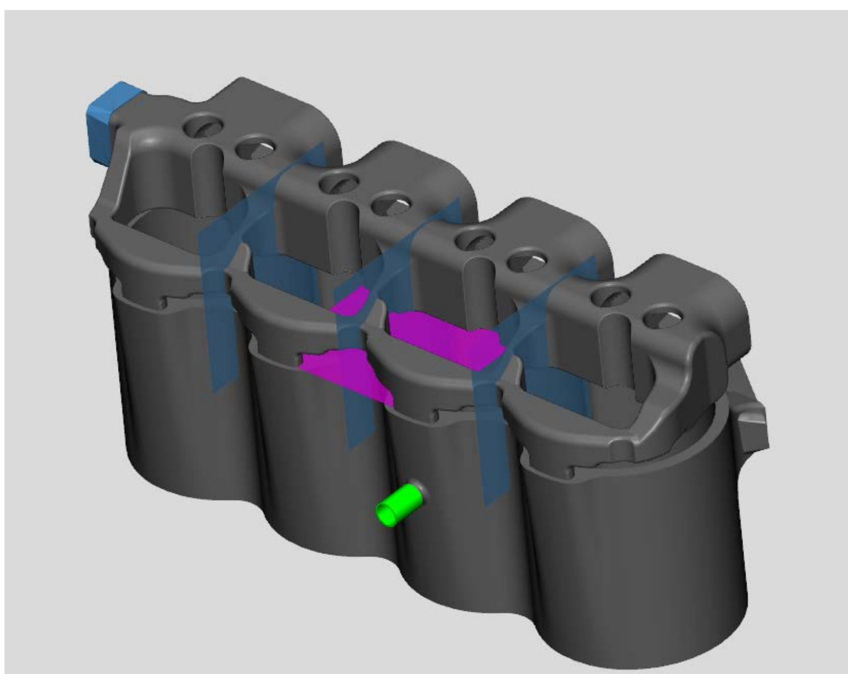
The **Datum Plane Size** attribute is for visual purposes. Regardless of the displayed size of the plane, the volume will be discretized as if the plane extends completely through the volume. Create two more child datum planes by copying and pasting the Jacket-1 plane, or using the Copy and Edit Object… right-click option. Update the locations of the planes so they divide between cylinders 2 and 3, and cylinders 3 and 4 as shown below.





Next, add the plane to divide the head and block water jackets as shown below. Note the change in the plane direction.


Template: DatumPlane
Help
Object Name: Jacket-4
Add Long Comment...
Object Comment:
Main
Visual
Attribute | Unit | Object Value
Location X (Global) | mm | 0
Location Y (Global) | mm | -213.5
Location Z (Global) | mm | 240.5
Direction X | | 0
Direction Y | | 0
Direction Z | | 1
Use as Discretization Plane | | ☑
Datum Plane Size | mm | 100
OK
Cancel
Apply



1.2.3 Adding External Connections

To define the inlet and outlet ports in the model, additional connections are needed. GEM3D will automatically create the connections in the model between flow volumes, and will assign subassembly connections to any flow ports which are not otherwise connected. However, the inlet and outlet directions are arbitrarily determined by GEM3D and may not match the desired direction. By defining connection directions, the inlet and outlet flow ports will be enforced.

To add a connection, right-click on the Inlet-Pipe component and select "Add Connection…" from the menu. Select the port which corresponds to the inlet end of the pipe (Port 1), and choose to create a new GEMSubAssExtConn. For the Port ID, enter a value of 1, and select Inlet from the Flow Direction drop-down menu. Click OK to create the connection.

Attribute	Unit	Object Value
Port ID		1
Flow Direction		Inlet

Repeat this process with the Head-Outlet pipe, only selecting Port 2 of the pipe instead of Port 1. For the Port ID number in the GEMSubAssExtConn SAconn2, enter a value of 2, and select Outlet from the Flow Direction drop-down menu.

Attribute	Unit	Object Value
Port ID		2
Flow Direction		Outlet

For the EGR-Outlet pipe, Port 1 should be selected for the GEMSubAssExtConn. For the Port ID number in the GEMSubAssExtConn SAconn3, enter a value of 3, and select Outlet from the Flow Direction drop-down menu.

Attribute	Unit	Object Value
Port ID		3
Flow Direction		Outlet

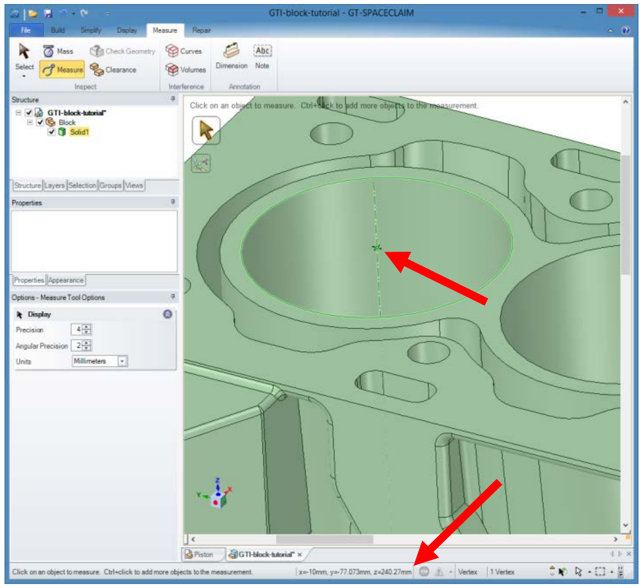
At this time, the flow model is complete in GEM3D. Save the model as water-jackets-tutorial.gem. For comparison, the water-jackets-tutorial-final.gem file is available in the tutorial directory, although this final file also includes the cylinder structure from the next two sections.

1.3 Creating Cylinder Structure Parts in GEM3D

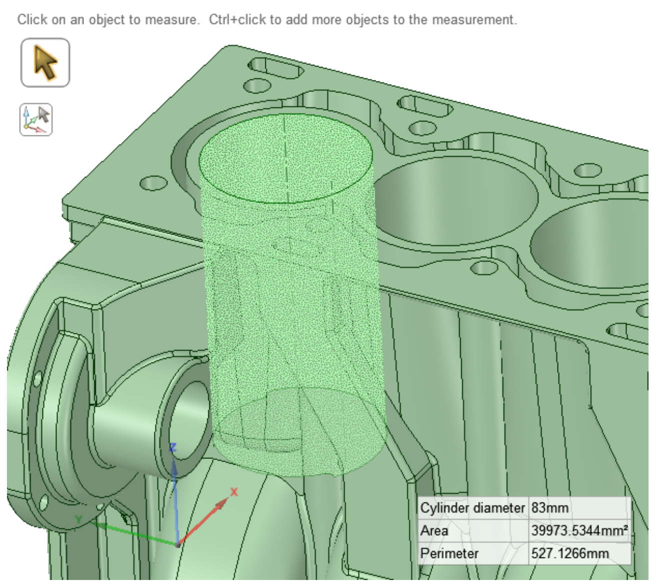
GT-SUITE includes a representative cylinder, head, piston, and port structure which can be used as the link between a combustion simulation and a cooling system simulation. This structure is created parametrically from user inputs, and uses a finite element method for calculating the structure temperature and resulting heat transfer rates. The EngCylTWallSoln template is commonly used in stand-alone engine simulations to provide realistic boundary conditions for the combustion calculation. The EngCylStrucCond template is used in either stand-alone cooling system simulations, or in coupled simulations between the engine and coolant system. This EngCylStrucCond template can also be created in GEM3D to easily match up the external surfaces with the water jacket geometry.

1.3.1 EngCylStrucCond Dimensions

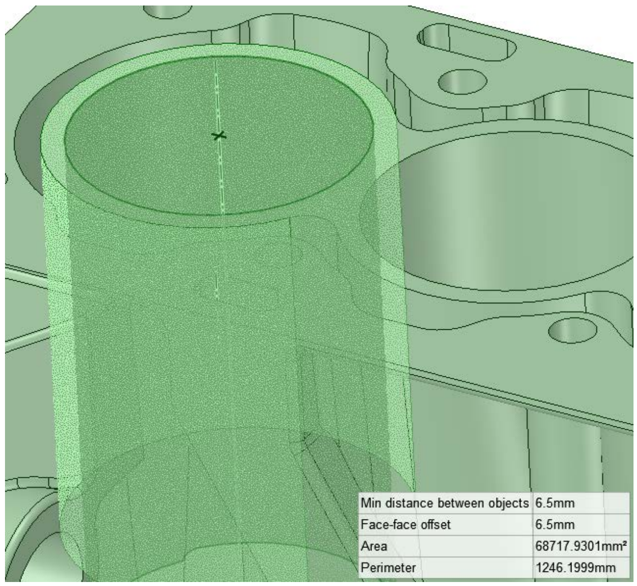
The EngCylStrucCond is created from basic part dimensions, which can be measured using GT-SPACECLAIM. Open the ACIS file GTI-block-tutorial.sab from the \tutorials\Modeling_Applications\Cooling_Engine\ directory in GT-SPACECLAIM. The first cylinder to be created is the cylinder closest to the water jacket entrance, Cylinder 1. The first dimension to be measured is the Cylinder location. This location is defined from the center of the cylinder bore, at the top of the cylinder surface. Select the Measure tool from the Measure menu, and move the mouse over the cylinder surface. A point should appear in the center of the cylinder bore, click on it and the location of the point will be given in the status bar.



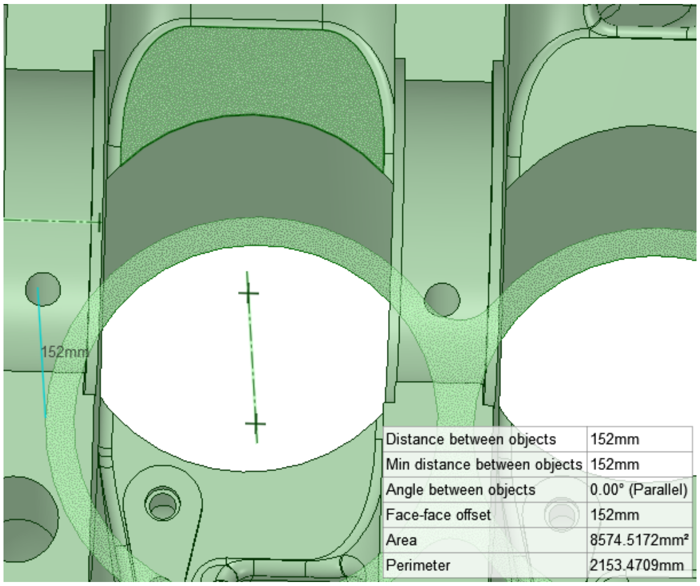
The location given for the point is (-10, -77.073, 240.27) mm. The next step is to measure the cylinder bore. Select the surface of the cylinder using the measure tool to obtain the diameter.



The measured cylinder bore diameter is 83 mm. Next, measure the wall thickness. To do this, click both the inside and outside surface of the cylinder wall while holding down the CTRL key. The Measure tool will give the difference between the two surfaces, which is the wall thickness.

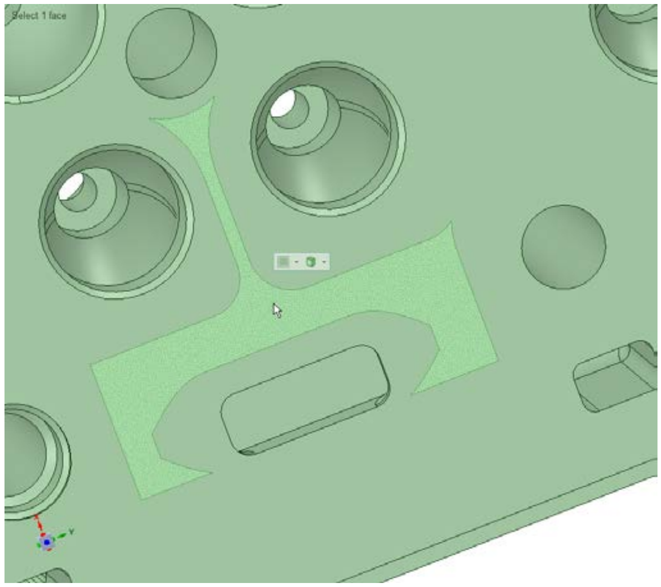


The wall thickness is measured at 6.5 mm. The next measurement is the cylinder length. This is measured from the top of the cylinder to the bottom, along the cylinder axis. Using the Measure tool, select the top surface of the cylinder. Then, while holding the CTRL key, select the flat surface at the base of the cylinder inside the crankcase. The measure tool will determine the distance between the surfaces, which measures 152 mm.

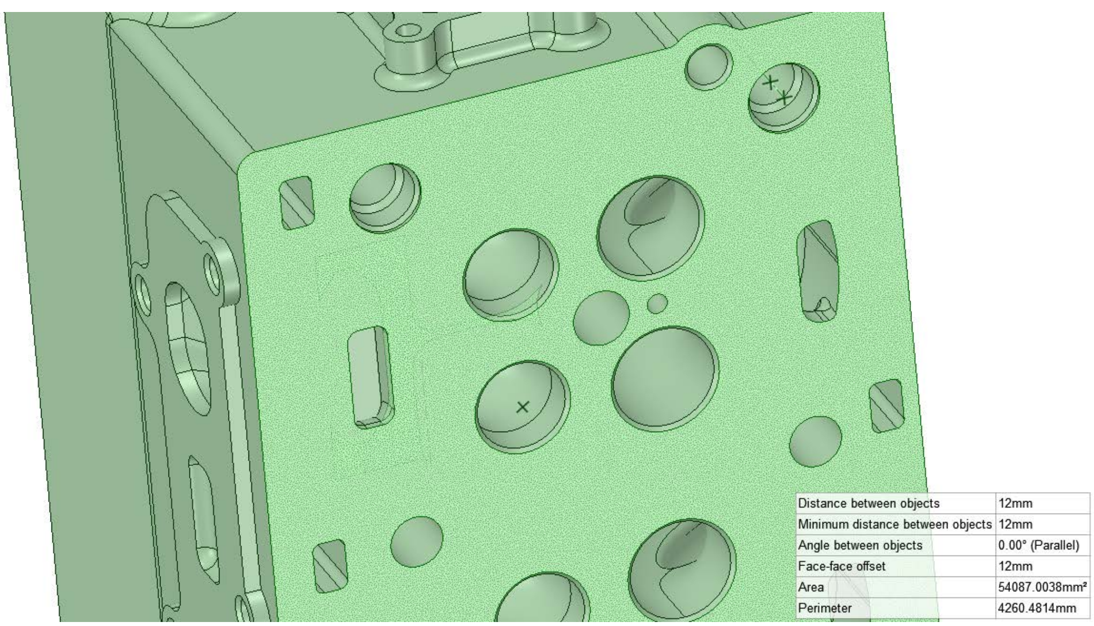


The next two dimensions are the head thickness and the head dome height. These will be determined from the head geometry, so open up the file named GTI-cylhead-tutorial.sab located in \tutorials\Modeling_Applications\Cooling_Engine\ using GT-SPACECLAIM. Because the bottom of the head is flat, the head dome height will be 0 mm. The head thickness will be measured from the bottom of the head to the base of the head coolant passages. This measurement is a somewhat arbitrary value, determining the boundary between the finite element solution for the structure temperature and a more simplified representation of the rest of the head. For more complicated water jacket shapes, this value can be estimated from the minimum distance from the combustion chamber to the head coolant passages.

To select the base of the head coolant passages, use a GT-SPACECLAIM feature called Query Select. Query Select allows the selection of surfaces which are "behind" other surfaces or objects. To activate query select, position the mouse over the bottom surface of the head, in between the exhaust valves and the inlet to the water jacket. The exhaust valves are the smaller diameter valves, on the -X side of the head. Next, hold the Control key and scroll the mouse wheel up. This will highlight the inner surface of the water jacket, as shown. Release the Control key, and left-click on the surface to select it.



Next, select the lower surface of the head while also holding the Control key to add to the current selection. Last, select the Measure tool to obtain the distance between the two surfaces. The measured distance should be 12 mm, which will be the head thickness.



1.3.2 Creating the EngCylStrucCond Component

From the measured dimensions, the EngCylStrucCond component can be created. Back in the GEM3D model that contains the water jackets, create a new EngCylStrucCond from the toolbar or the Builder > Add Component menu. For the object name, use "Cylinder1". This cylinder will serve as a template for the other cylinders in the model, with the other cylinders requiring changes to the location and water jacket dimensions.

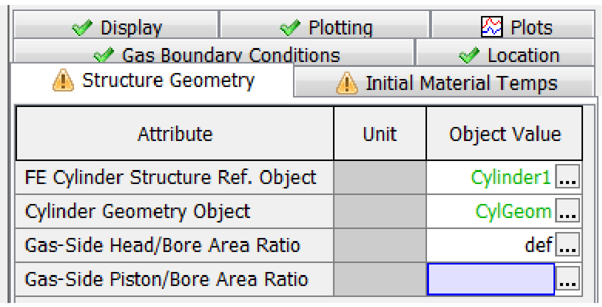
The first folder in the EngCylStrucCond contains the geometry reference objects. The FE Cylinder Structure Ref. Object defines the finite element mesh, using the dimensions measured above. Create a new reference object named "Cylinder-1" for this attribute. A new reference object will be created for each cylinder due to the different water jacket dimensions, though most of the attributes will remain the same.

The next attribute is the Cylinder Geometry Object. Because a corresponding GT-POWER model is available for this engine (Diesel_Engine_Tutorial.gtm, available in this tutorial directory), the cylinder geometry values from that engine will be used. Create a new reference object called "CylGeom". Double-click on this reference object to create a new EngCylGeom object, and enter the measured bore diameter of 83 mm (measured from the previous tutorial section). The remaining attributes will be entered from the GT-POWER model: The Stroke is 92.4 mm, the Connecting Rod Length is 129 mm, the Compression Ratio is 17, and The TDC Clearance Height is 1 mm. When complete, the CylGeom reference object should appear as shown:

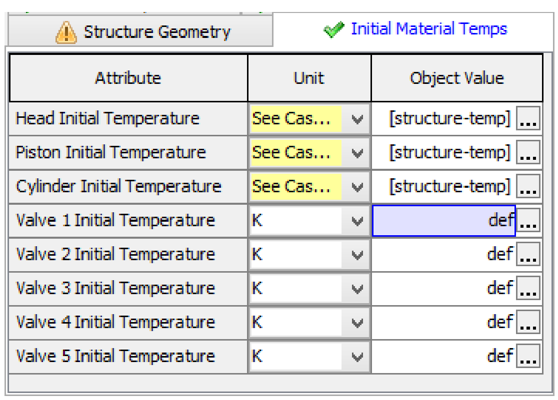
Main | Piston-to-Crank Offset | Crank-Slider Compliance

Attribute	Unit	Object Value
Bore	mm	83
Stroke	mm	92.4
Connecting Rod Length	mm	129
Compression Ratio		17
TDC Clearance Height	mm	1

The last two attributes are the Gas-Side Head and Gas-Side Piston Area Ratios. Because the bottom of the cylinder head is flat, the Gas-Side Head/Bore Area Ratio should be "def". For the piston attribute, the value can be determined from a measurement of the geometry. This measurement will be taken along with the rest of the piston dimensions. For now, leave the attribute blank as shown below.



The next folder in the EngCylStrucCond defines the initial temperatures. Create a new parameter, which will also be used in the thermal masses in this model for the initial temperatures. This parameter will make it easier later on to switch the model from the calibration run to the full model run. For the head, piston, and cylinder initial temperature attributes use the parameter "[structure-temp]". The folder should appear as shown:



Back in the Structure Geometry folder, double-click on the "Cylinder-1" reference object to create a new 'FECylinderStructure' object. The first folder is the Structure-Structure HTR folder. This folder defines the values for the conduction between parts of the cylinder structure. Some default heat transfer coefficients are included; these should only be adjusted for calibrating the model or if a heat transfer coefficient or contact resistance is known from experiments. For the Head Gasket Material, use a value of CarbonSteel for this model. This pre-defined material can be selected from the GT Library by using the Value Selector button in the object window. Choose CarbonSteel from the list, and click OK to fill out the field. For the Head Gasket Thickness, use a value of 2 mm. The next two attributes, "Ring/Engine Friction Ratio" and "Skirt/Engine Friction Ratio," define the heat generated by friction between the piston skirt/rings and the cylinder liner. These attributes define the fraction of the cylinder friction attributed to the ring-cylinder and skirt-cylinder interfaces. The friction heat is imposed as a heat rate on the surface FE nodes of the piston rings and skirt. For the Ring/Engine Friction Ratio and Skirt/Engine Friction Ratio, the values should be 0.5. The default values are appropriate if the friction is being automatically obtained from a total FMEP value, but for this model the cylinder friction will be directly imposed.

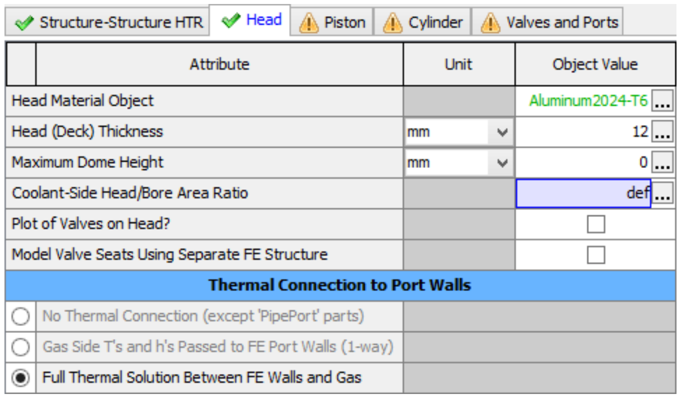
The friction heat must be supplied to the cylinder structure at the port labeled "Cylinder (Ring and Skirt) Friction." This quantity may be supplied via a 'ThermalCompConn' connection part connected to either an 'EngineCrankTrain' part (when an engine model is present) or to an 'EngineState' part (for a mapped engine). Please be aware that if the friction model in the engine is the 'EngFrictionDetail' model, the friction heat will include ONLY the friction attributed to the cylinder and will NOT include the friction attributed to the crankshaft or valvetrain. In this case, the two fractions should sum to 1.0. If the friction model in the engine is not modeled using the 'EngFrictionDetail' object, then the friction heat will included the full engine friction (including the crankshaft and valvetrain) and smaller fractions will be appropriate.

Another approach to modeling the friction is to calculate the friction values using controls components, and impose the friction heat on the cylinder structure using the HeatRate component. For a further discussion on engine friction calculations, see Chapter 1.10. The completed folder should match the figure below:

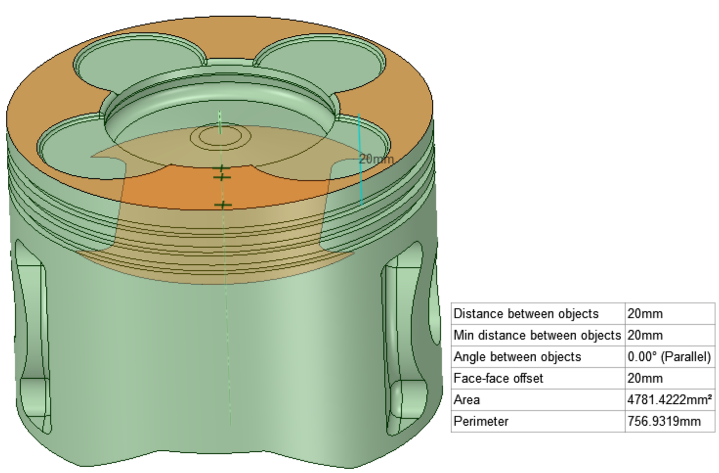
Structure-Structure HTR | Head | Piston | Cylinder | Valves and Ports

Attribute	Unit	Object Value
Skirt to Cylinder HTR Coefficient	W/(m^2-K)	def (=5000.0)
Ring to Piston HTR Coefficient	W/(m^2-K)	def (=15000.0)
Ring to Cylinder HTR Coefficient	W/(m^2-K)	def (=30000.0)
Valve to Seat HTR Coefficient	W/(m^2-K)	def (=50000.0)
Valve to Guide HTR Coefficient	W/(m^2-K)	def (=12500.0)
Head Gasket Contact Resistance	(m^2-K)/W	def (=5.0E-5)
Head Gasket Material Object		CarbonSteel
Head Gasket Thickness	mm	2
Ring/Engine Friction Ratio		0.5
Skirt/Engine Friction Ratio		0.5
Head to Valve Seat Contact Resistance	(m^2-K)/W	def (=1.0E-12)

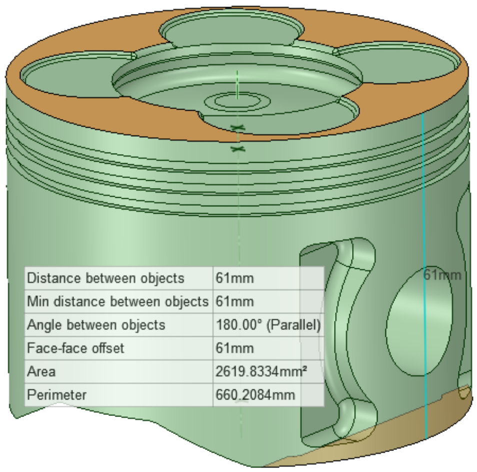
The next folder is the Head folder, which describes the geometry of the head immediately surrounding the combustion chamber. For the Head Material Object, select Aluminum2024-T6 from the list available in the Value Selector button .... Aluminum2024-T6 will be available in the GT-SUITE library because it has not yet been imported to the current model. For the Head (Deck) Thickness attribute, use the value of 12 mm as measured in the previous section. Because the combustion chamber of this particular engine is not domed, use a value for the Maximum Dome Height of 0 mm. The Coolant-Side Head/Bore Area Ratio attribute should be given a value of "def". This indicates that the surface area on the coolant side of the head disc will be calculated from the finite element geometry directly based on the geometrical inputs by the user. The last set of attributes in the folder are radio buttons which define the thermal connection between the ports and the head mesh in an engine (or integrated) model. Select the third option, so the full boundary conditions will be passed between the gas and thermal circuits in the integrated model. The completed Head folder should match the figure below:



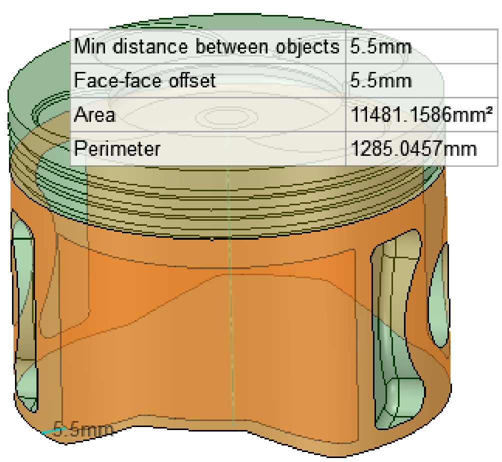
The next geometry to define is the Piston. The piston geometry for this tutorial is located in the ACIS file GTI-piston.sab. Open the geometry in GT-SPACECLAIM and record the measurements as discussed below. For the Piston Material Object, select CarbonSteel from the GT-SUITE library using the Value Selector button . The Piston Top (Deck) Thickness can be measured from the geometry in GT-SPACECLAIM by selecting the top surface of the piston, selecting the underside of the piston while holding the Ctrl key, and then selecting the Measure tool from the Measure menu. The distance should be measured as 20 mm:



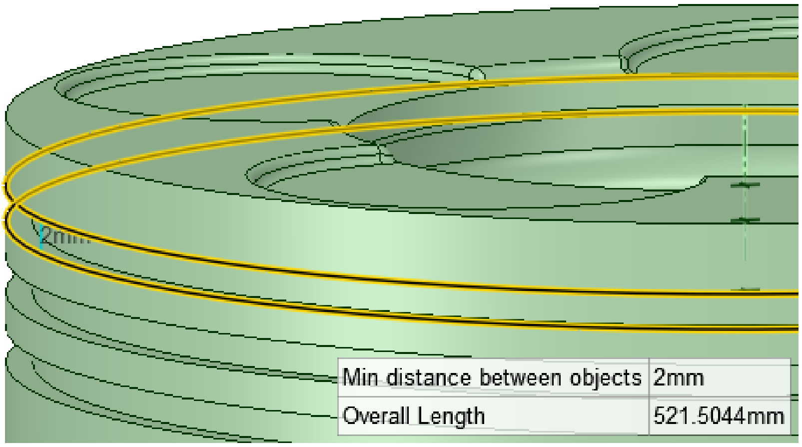
For the Piston Cup Object, give a reference object name of "piston-cup". This reference object will be defined once the other piston geometry has been entered. The piston height is measured from the top surface to the bottom surface of the piston, 61 mm, as shown below:



The next dimension to measure is the Skirt Thickness, which is determined from the distance between the outside and inside surface of the piston skirt. This distance is 5.5 mm.

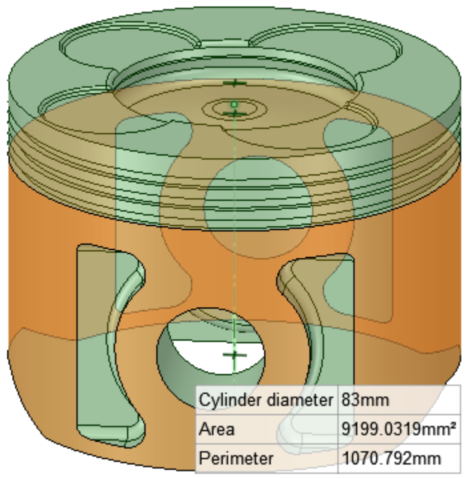


The ring thickness is determined from the groove size in the piston. This distance is 2 mm.

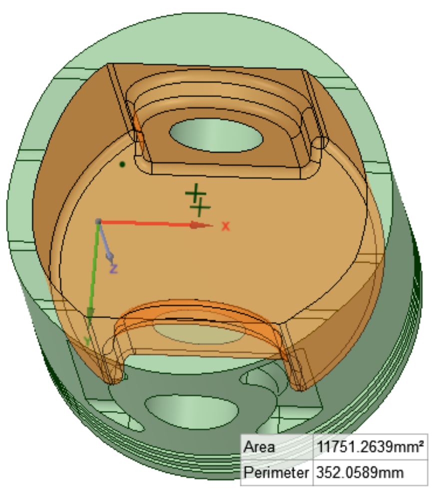


For the following attributes, using "def" means that the values are obtained automatically from the generated finite element structure. This is fine at the early project stages if there is no other data available. However, since most piston geometries are not "general" it is recommended to fill out the attributes based on the real geometry. This will correct the heat transfer areas on the generated piston to match the actual surface areas.

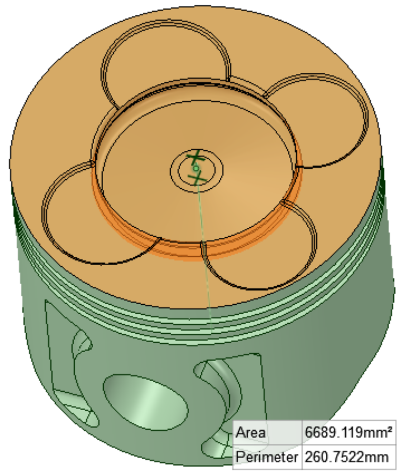
The next attribute is the Normalized Effective Skirt Area. This is defined by the ratio of the actual contact area between the skirt and the wall to the theoretical contact area. The theoretical area is determined from the formula = π * Bore Diameter * (Piston Height - Piston Top (Deck) Thickness). Measure the outside area of the piston, and then divide it by 10690.84 mm^2. Based on a piston skirt area of 9199.03, this ratio should be .86.



The Underside Piston/Bore Area Ratio is used to determine the surface area of the piston which is in contact with the oil. To determine this ratio, measure the surface area on the geometry and divide it by the bore area (= π * Bore Diameter^2 / 4). Based on the GT-SPACECLAIM measurement, the ratio is 2.17.



The Gas-Side Piston/Bore Area Ratio is calculated using a similar formula, only using the top side of the piston instead of the underside. The ratio for this geometry is 1.236. This ratio should be entered in the "Structure Geometry" folder of the EngCylStrucCond, as discussed previously.

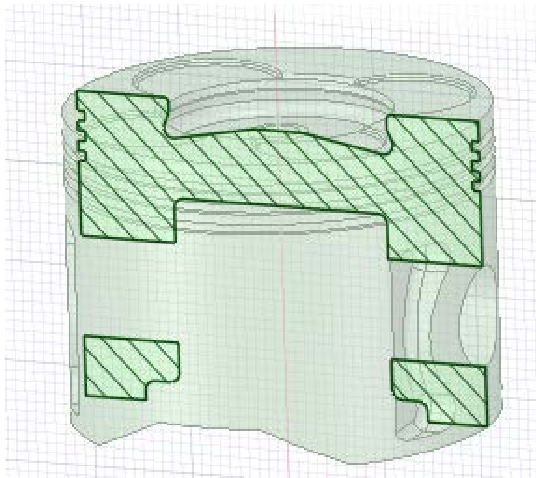


The last attribute for the folder is the piston volume. This is measured in GT-SPACECLAIM by selecting the complete solid and using the Mass tool. The measured volume for this piston is 160634.1 mm^3. When the piston folder is completed, it should appear as shown below:

Structure-Structure HTR | Head | Piston | Cylinder | Valves and Ports

	Attribute	Unit	Object Value
Piston Material Object			CarbonSteel
◉	Piston Top (Deck) Thickness	mm	20
	Piston Cup Object		piston-cup
	Piston Height	mm	61
	Skirt Thickness	mm	5.5
	Ring Thickness	mm	2
	Normalized Effective Skirt Area		0.86
	Underside Piston/Bore Area Ratio		2.17
	Piston Volume	mm^3	160634.1
○	Custom FE Piston Reference Object		

To finish up the piston geometry, the piston-cup reference object that was named needs to be completed. Double-click on the name to open the EngCylPistonCup reference object. These dimensions can be determined by measurements made on a cross section of the piston. To view a cross section, move the mouse over the base of the piston bowl until the center axis of the piston appears. Then, select this axis and choose the Section Mode button from the Build menu. The piston cross section should appear:



The Piston Cup Diameter (Maximum) can be measured from the distance between the rounds at the base of the bowl, plus the diameter of the round. This total diameter is 47.4 mm. The Maximum Piston Cup Depth is measured from the top surface of the piston to the bottom of the round at the base of the bowl. The Measure tool will give the distance between the top surface and the center of the round, adding the round radius results in a distance of 8.05 mm. The Piston Cup Diameter (Edge) is determined from the diameter of the cylindrical surface at the top of the bowl, which is 45 mm. The last measurement is the Piston Cup Center Depth, measured as 5 mm.

Help Object: piston-cup Add Long Comment...

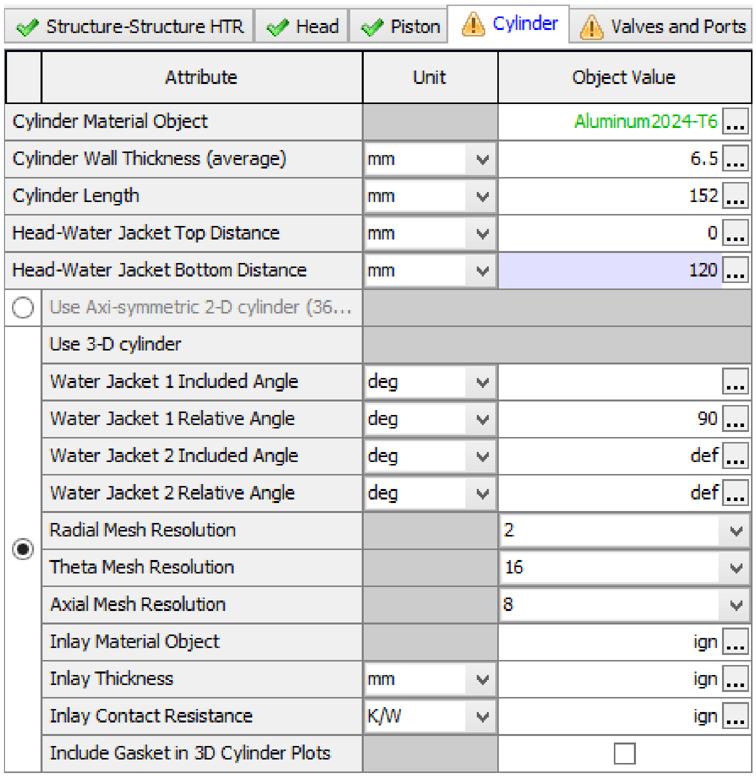
Comment:

Main

Attribute	Unit	Object Value
Piston Cup Diameter (Maximum)	mm	47.7
Piston Cup Depth at Maximum Diameter	mm	8.05
Piston Cup Diameter (Edge)	mm	45
Piston Cup Center Depth	mm	5

OK Cancel Apply

The next folder is the Cylinder definition. For the Cylinder Material Object, use the value selector to choose the Aluminum2024-T6 material. The Cylinder Wall Thickness was measured in the previous section as 6.5 mm, and the Cylinder Length was measured to be 152 mm. Because the top of the water jackets are even with the top surface of the block, the Head-Water Jacket Top Distance is 0 mm. The water jackets extend 120 mm from top to bottom (this can be measured in GT-SPACECLAIM), so enter 120 mm for the Head-Water Jacket Bottom Distance since the top is at 0 mm. The water jacket angles typically require some manual matching to the geometry. For now, leave this section with the default values; it will be updated later. The Cylinder folder should appear as shown:



Attribute	Unit	Object Value
Cylinder Material Object		Aluminum2024-T6
Cylinder Wall Thickness (average)	mm	6.5
Cylinder Length	mm	152
Head-Water Jacket Top Distance	mm	0
Head-Water Jacket Bottom Distance	mm	120
Use Axi-symmetric 2-D cylinder (36...		
Use 3-D cylinder		
Water Jacket 1 Included Angle	deg	
Water Jacket 1 Relative Angle	deg	90
Water Jacket 2 Included Angle	deg	def
Water Jacket 2 Relative Angle	deg	def
Radial Mesh Resolution		2
Theta Mesh Resolution		16
Axial Mesh Resolution		8
Inlay Material Object		ign
Inlay Thickness	mm	ign
Inlay Contact Resistance	K/W	ign
Include Gasket in 3D Cylinder Plots		

The Valves and Ports folder will be filled out once the EngCylStrucCond is correctly positioned. This allows the valves to be lined up with the 3D geometry, so skip this folder for now. When the folders have been filled out as described above, click **OK** to complete the cylinder structure reference object.

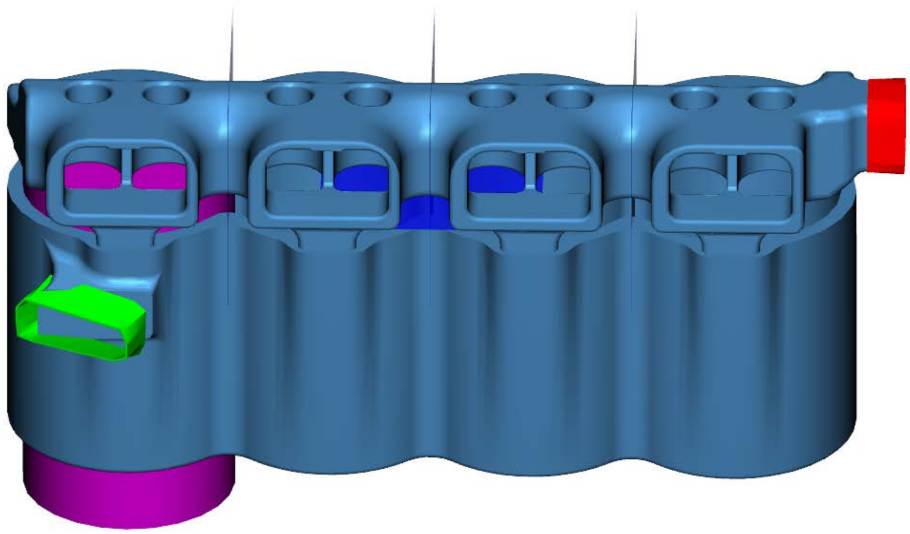
Back in the Cylinder1 (EngCylStrucCond) object, the Location folder is used to give the absolute location and orientation of the cylinder in 3D space. The cylinder location was measured at a position of (-10, -77.073, 240.27) in GT-SPACECLAIM. The cylinder location is not critical to the model solution, but defining it has two advantages. The first is that the cylinder can be matched up to the water jackets in

GEM3D, which makes defining the water jackets easier. The second is that if multiple cylinders are plotted in the same view in GT-Post, defining the cylinder location for each part will prevent the meshes from overlapping. The Direction vector refers to the direction of the cylinder bore from the bottom of the cylinder to the top (i.e., pointing from the crankshaft toward the head). For this inline engine, use a vector of (0, 0, 1). Other engine types (V, Flat) will require measuring the direction of the cylinder axis. The Crankshaft Center Line direction is used to determine the relative water jacket and valve locations. This direction is typically defined from the front (accessory) end to the back (transmission) end of the engine. For this engine orientation, a vector of (0, -1, 0) should be used.

Structure Geometry | Initial Material Temps | Gas Boundary Conditions | Location | Display | Plotting

Attribute	Unit	Object Value
Location X	mm	-10
Location Y	mm	-77.073
Location Z	mm	240.27
Direction X		0
Direction Y		0
Direction Z		1
Crankshaft Center Line Direction X		0
Crankshaft Center Line Direction Y		-1
Crankshaft Center Line Direction Z		0

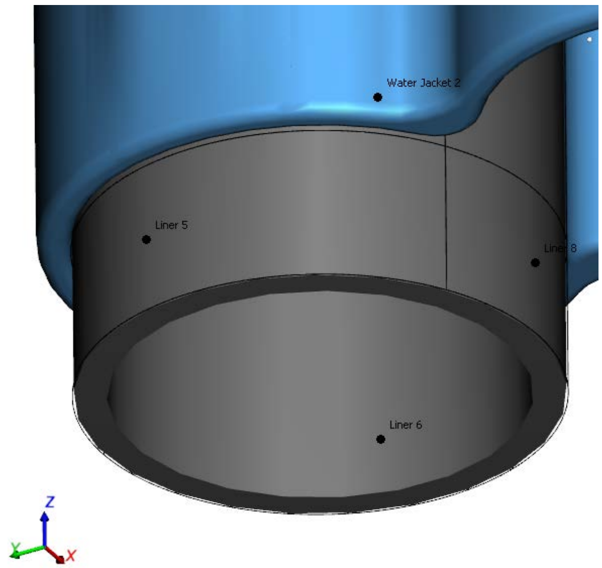
Once the location and directions have been defined, the EngCylStrucCond can be displayed in GEM3D. Click the OK button and the component will appear as shown:



1.3.3 Water Jacket Definitions

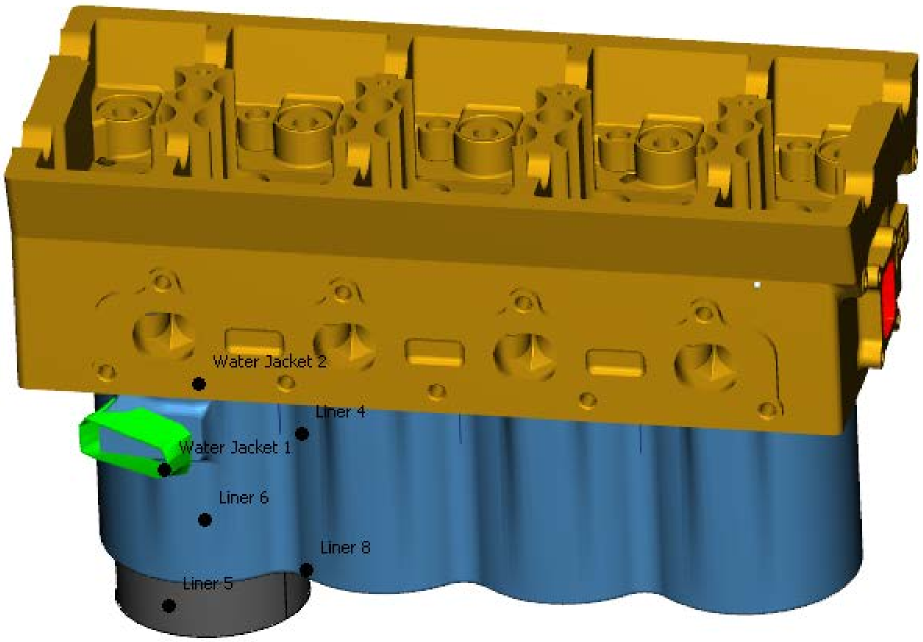
The next step is to position the water jackets on the cylinder. The 3D locations of the water jackets are important to capture the temperature distribution on the cylinder liner. The water jacket included and relative angles should be matched up to the water jacket geometry such that the labeled surfaces on the cylinder are in contact with the flow volume. The angles, along with the distances defined in this folder, will capture the correct heat transfer areas and their locations. Open up the Cylinder-1 (FECylinderStructure) reference object, and navigate to the Cylinder folder. For this particular geometry, the Water Jacket 1 Included Angle is 150 degrees, and the relative angle is 105 mm. Because the block water jackets are symmetric across the crankshaft direction, the values for the water jacket 2 angles can be "def". The resulting water jackets for Cylinder 1 are shown below, the water jackets touch on the +Y side, and are open on the -Y side to match the water jacket geometry.

Water Jacket 1 Included Angle	deg	150
Water Jacket 1 Relative Angle	deg	105
Water Jacket 2 Included Angle	deg	def
Water Jacket 2 Relative Angle	deg	def



1.3.4 Valve and Port Definitions

The last step in creating the EngCylStrucCond is to define the valves and ports. To assist with the measurement and location of the ports, import the GTI-cylhead-tutorial.sab ACIS file into the water-jackets-tutorial.gem file. The model should appear as shown.



The diameter of the intake and exhaust valve faces, if they are unknown, can be measured from the geometry using the Dimensions > Measure Distance tool. This tool will measure the distance and component distances between two points placed on the model.

The X and Y Coordinates of the Valve Center are defined using each cylinder's local coordinate system. To view the origin for a cylinder, right click on the cylinder in the 3D window and select "Component's Axis." Measure the distance from the component's axis to the approximate center of each valve, and enter those dimensions in for each valve's column. After the approximate face diameter and center coordinates have been entered, they can be fine-tuned visually, by viewing the cylinder structure from the bottom and matching up the dimension to the head solid shape. The completed dimensions for this geometry are shown below.

Structure-Structure HTR | Head | Piston | Cylinder | Valves and Ports

Attribute	Unit	Valve #1	Valve #2	Valve #3	Valve #4
Valve/Port Number		1	2	3	4
Valve Face Diameter	mm	26	26	23	23
Valve Head Thickness	mm	def	def	def	def
X-Coordinate of Valve Center	mm	-20	20	-18	18
Y-Coordinate of Valve Center	mm	17	17	-18.5	-18.5

The remaining materials and dimensions can be estimated or filled out from known engine specifications. Here is the completed Valves and Ports folder: Note that the Valve Open Fraction, Port Length, and Port Diameter are entered directly in to the structure geometry. In an engine or integrated model, these attributes should be set to "def" so that the values can be taken from the connected pipe parts.

Structure-Structure HTR | Head | Piston | Cylinder | Valves and Ports

Attribute	Unit	Valve #1	Valve #2	Valve #3	Valve #4
Valve/Port Number		1	2	3	4
Valve Face Diameter	mm	26	26	23	23
Valve Head Thickness	mm	def	def	def	def
X-Coordinate of Valve Center	mm	-20	20	-18	18
Y-Coordinate of Valve Center	mm	17	17	-18.5	-18.5
Valve Back Heat Transfer Multiplier		def (=1.0)	def	def	def
Valve Material Object		CarbonSteel	CarbonSteel	CarbonSteel	CarbonSteel
Valve Open (Time) Fraction		0.3	0.3	0.3	0.3
Port Length	mm	85	85	95	95
Port Diameter	mm	25	25	22	22
Port Thickness	mm	4	4	2	2
Valve Cavity		ign	ign	ign	ign
Valve Seat Dimensions		def	def	def	def
Valve Guide Dimensions		ign	ign	ign	ign
Angled Back-Cut on Valve Head		☑	☑	☑	☑

1.3.5 Creating Other Cylinders

With Cylinder1 created, the other cylinders can be added. Copy Cylinder1 and paste three more cylinders into the model. Most of the definitions for the other 3 cylinders will be the same, the main changes include the location and water jacket definitions. The new locations are:

Cylinder2: (-10, -168.073, 240.27)

Cylinder3: (-10, -259.073, 240.27)

Cylinder4: (-10, -350.073, 240.27)

The structure reference objects must also be modified to match the water jacket values. Copy the Cylinder-1 reference object and paste three more objects into the model. In the Cylinder folder of Cylinder-2, the water jacket definitions should also be changed to:

Water Jacket 1 Included Angle	deg	120
Water Jacket 1 Relative Angle	deg	90
Water Jacket 2 Included Angle	deg	def
Water Jacket 2 Relative Angle	deg	def

For Cylinder-3, the water jacket definitions should match Cylinder-2:

Water Jacket 1 Included Angle	deg	120
Water Jacket 1 Relative Angle	deg	90
Water Jacket 2 Included Angle	deg	def
Water Jacket 2 Relative Angle	deg	def

For Cylinder-4, the water jacket wraps around the other side of the cylinder from Cylinder-1, so the new angles are:

Water Jacket 1 Included Angle	deg	150
Water Jacket 1 Relative Angle	deg	75
Water Jacket 2 Included Angle	deg	def
Water Jacket 2 Relative Angle	deg	def

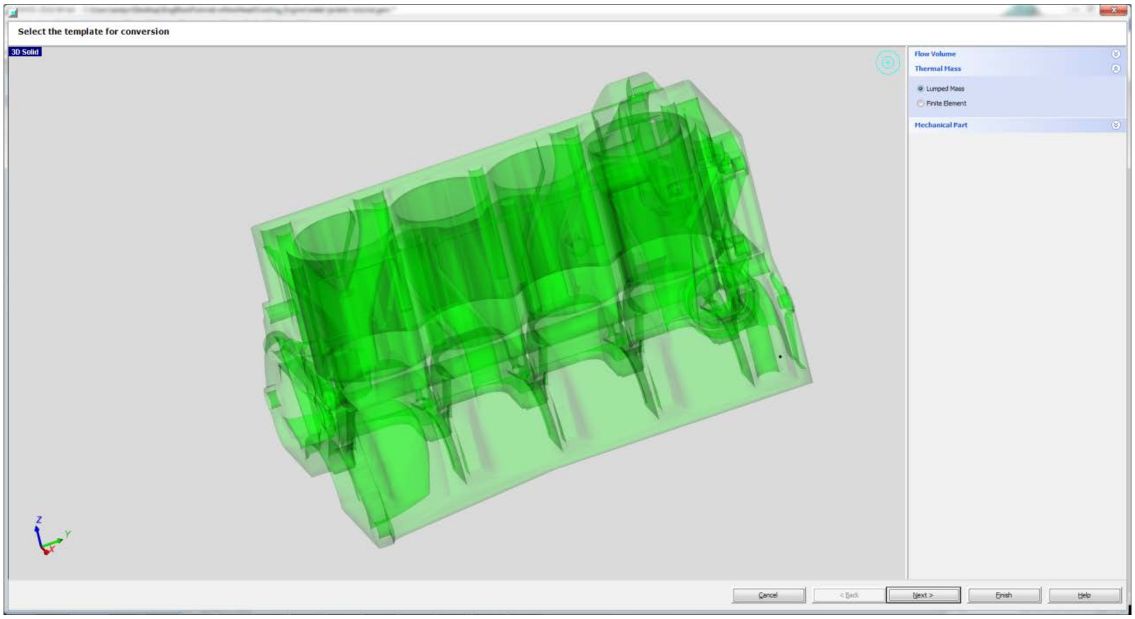
Finally, change the new EngCylStrucCond components to match the new FECylinderStructure reference objects (Cylinder2 EngCylStrucCond should reference the "Cylinder-2" FECylinderStructure, and so on). After hiding the solid shape of the cylinder head, the model should appear as shown.

1.4 Creating Thermal Masses for the Engine Block and Head

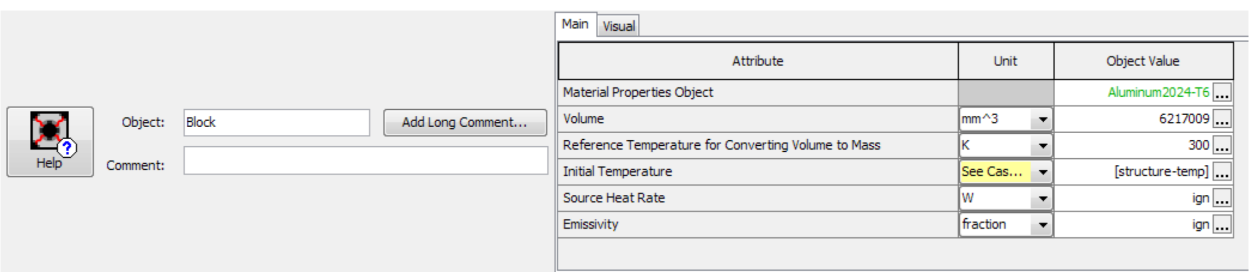
The EngCylStrucCond component contains the portion of the engine which will have the most significant temperature gradients, but it represents a small portion of the overall engine mass. For modeling the transient temperature change of the engine, it is important to include the rest of the engine block and head in the thermal model. These components will be converted from the original CAD geometry to obtain the mass and conduction paths.

1.4.1 Converting the Thermal Masses

Un-hide the imported cylinder head shape, and import the GTI-block-tutorial.sab file from the \tutorials\Modeling_Applications\Cooling_Engine\ directory. Select the block shape first, and choose the Convert Shape to Component tool from the right-click or Slicer menus.

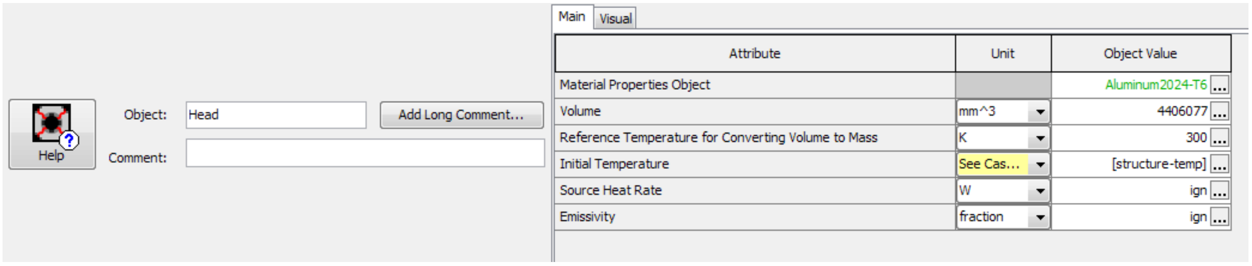


Select the Thermal Mass option from the right-hand side, select "Lumped Mass," and click Next. The conversion wizard will then calculate the volume of the Thermal Mass. On the next page, fill in the Object Name with "Block," and the Material Properties Object with Aluminum2024-T6. For the Initial Temperature, use the existing parameter "[structure-temp]":

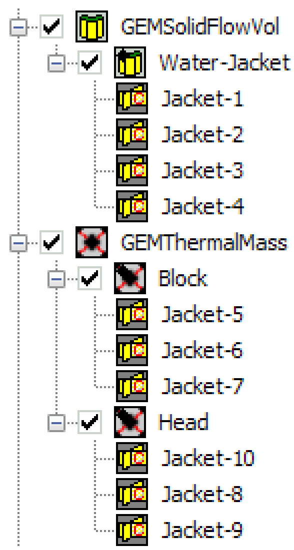


Attribute	Unit	Object Value
Material Properties Object		Aluminum2024-T6
Volume	mm^3	6217009
Reference Temperature for Converting Volume to Mass	K	300
Initial Temperature	See Cas...	[structure-temp]
Source Heat Rate	W	ign
Emissivity	fraction	ign

When the attributes have been entered, select Finish to complete the conversion. Repeat the steps for the head shape using the same Material and Initial Temperature, to create the "Head" GEMThermalMass:



Both the Head and Block masses will need to be divided into individual sections, in the same way the Water-Jacket GEMSolidFlowVol will be divided. Copy the Jacket-1, Jacket-2, and Jacket-3 child datum planes from the Water-Jacket, and paste them onto both the Block and Head GEMThermalMass components. When completed, the model tree should appear as shown:



1.4.2 Adding Ports to the Thermal Masses

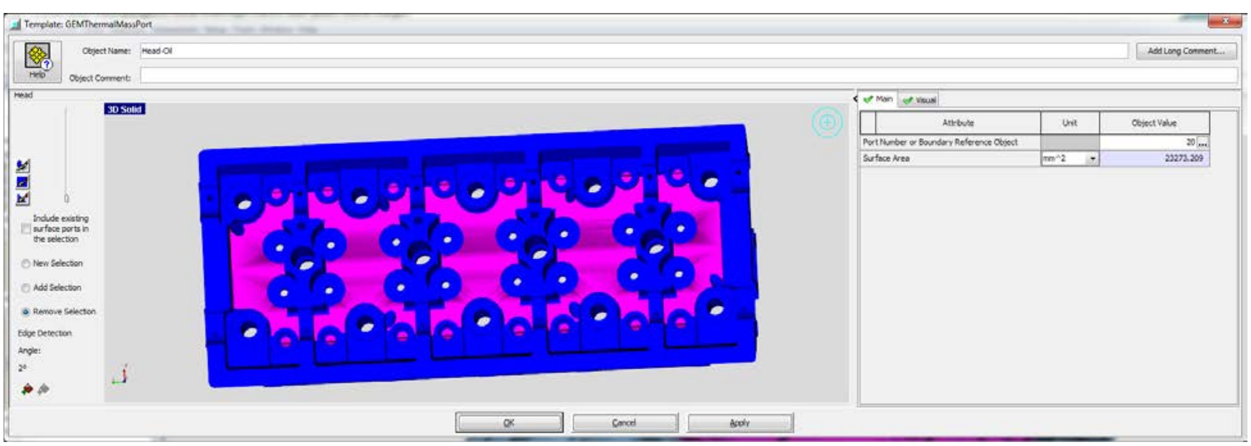
Once the Thermal Masses have been created and the discretization planes have been linked to each mass, additional connections to the masses can be defined in GEM3D. These connections will be used for linking the thermal masses to the environment and the oil boundary conditions, and will be measured using the actual surface areas from the solid geometry.

To create a new connection, right-click on the Head and select the "Thermal Mass Port…" option from the menu. This will open a new window where the surfaces can be selected. The first port to be created will be the connection to the oil. Set the Object name to Head-Oil. The selection of the port surfaces requires "marking" the appropriate areas on the geometry.

On the left hand side of the window, the vertical slider controls the Edge Detection Angle for selecting the surfaces. When a particular area is selected on the 3D model using a left-click, the adjacent surfaces to the selected area will be also selected depending on the Edge Detection Angle. If the neighboring surface has a contact angle less than the specified angle, it will be selected along with the original area. For now, drag the slider to set this angle to 5 degrees.

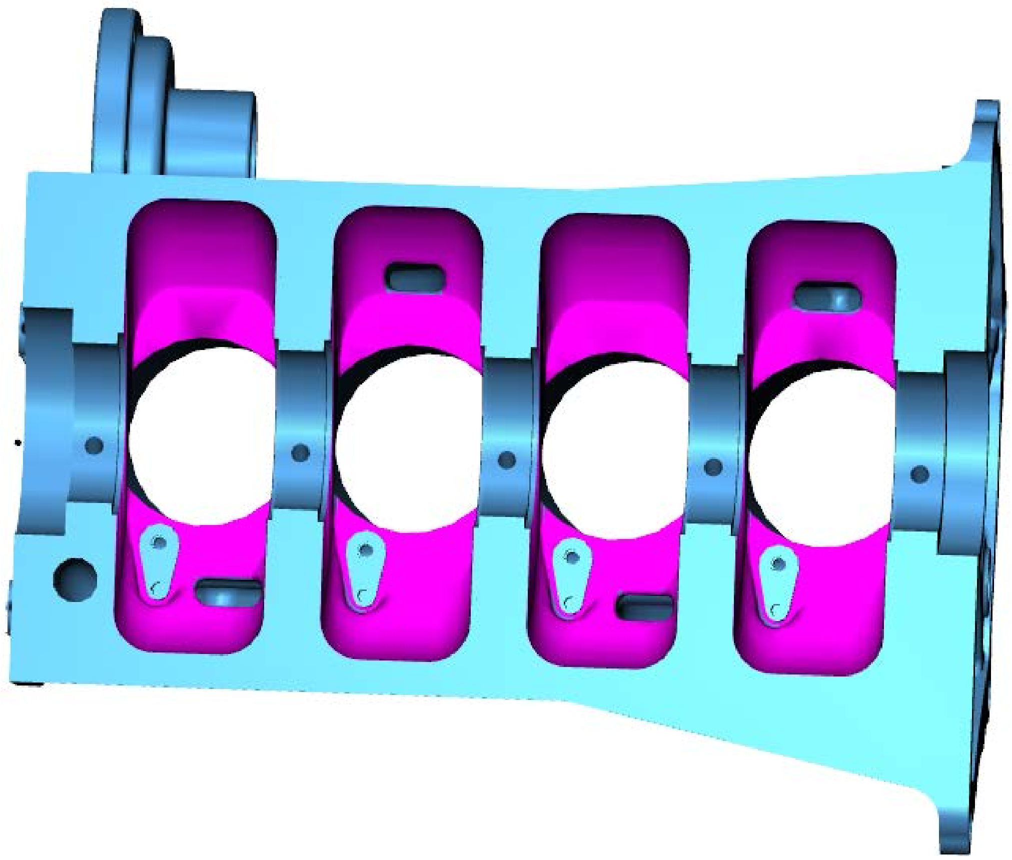
The radio buttons below the slider control the behavior of a the left-click on the 3D model: either the existing selection will be discarded and a new surface will be selected, or the existing selection will be kept and the new surfaces will be added to the selected area, or the new selection will be subtracted from the existing selected surfaces. Choose the Add Selection option.

In the 3D window, left click on the surface at the base of the head cavity where the oil will pool before returning to the pan. This will be used for the contact surface where heat transfer between the head and the oil volume occurs. Two or more selections may be needed to include the entire surface, and if any extra surfaces are included, use the Remove Selection option (with a smaller Edge Detection Angle) to de-select them. After completion, the selected surface should appear as shown below.

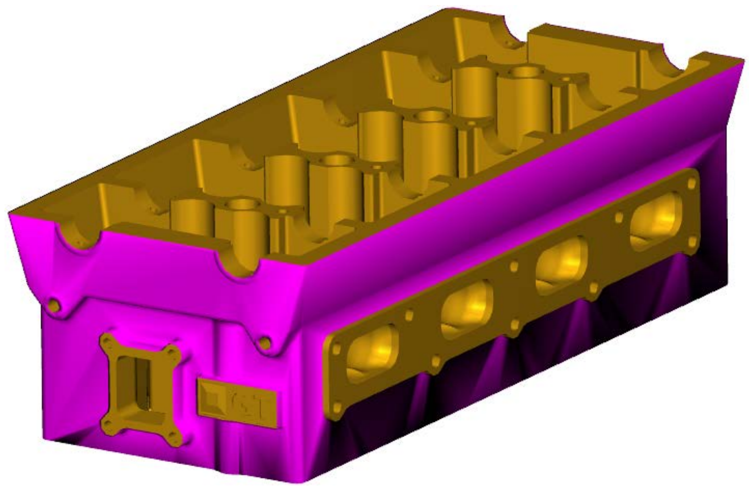


Note that when a new surface is selected, or the selection is updated, the Surface Area attribute is updated with the area of the selected surfaces. The last attribute to specify is the Port Number; in this case, set a port number of 20. This will be used for the port number of this connection even as the model is discretized and separate thermal masses are created. A consistent port number will make it easier to make the appropriate connections between the oil volumes and the thermal masses in GT-ISE. When the correct surface has been selected, the port number has been defined, and the object has been named, select OK to complete the port creation.

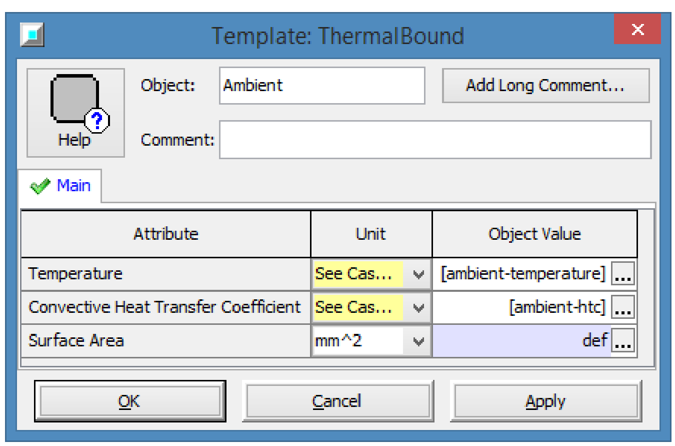
The next port to create is the oil port on the Block. Right-click on the Block GEMThermalMass, and select the "Thermal Mass Port…" command from the menu. Use a combination of the Add Selection and Remove Selection tools, varying the Edge Detection Angle, until the surfaces on the underside of the block are selected as shown below. Note that a smaller Edge Detection Angle makes it easier to control the surfaces that are selected, but will require more clicks to get the exact surface. The approximate surface area is 137295 mm^2, but the actual surface area will vary depending on what exact surfaces are selected. Set the Port Number to 20 to match the Head-Oil port number, and name the object Block-Oil.



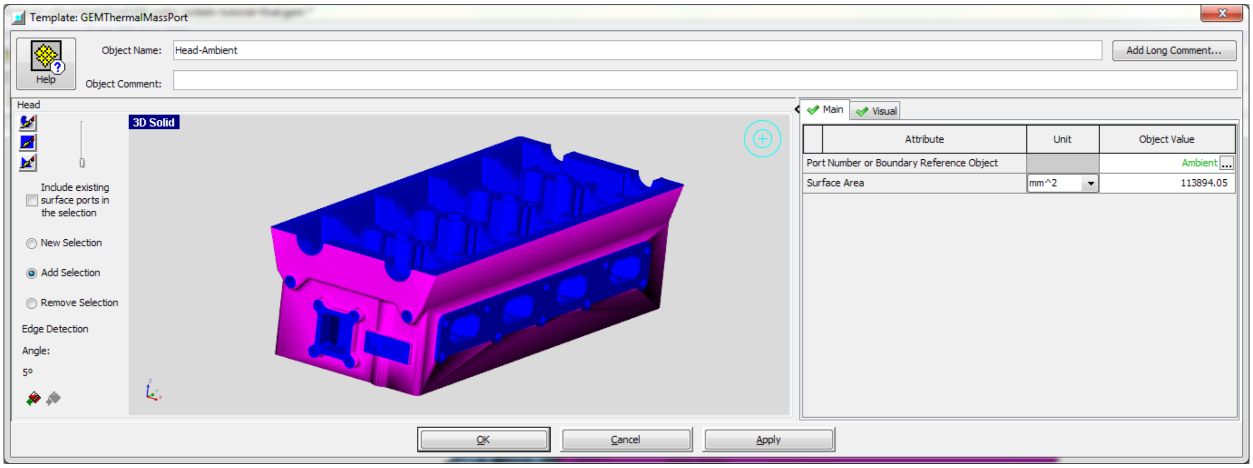
The next port to create is the port for the connection to the environment on the Head. Right-click on the Head and select the "Thermal Mass Port…" option from the menu. Using an edge detection angle of 50 degrees, mark the surfaces as shown below. The total surface area should be 113894 mm^2 (approximately).



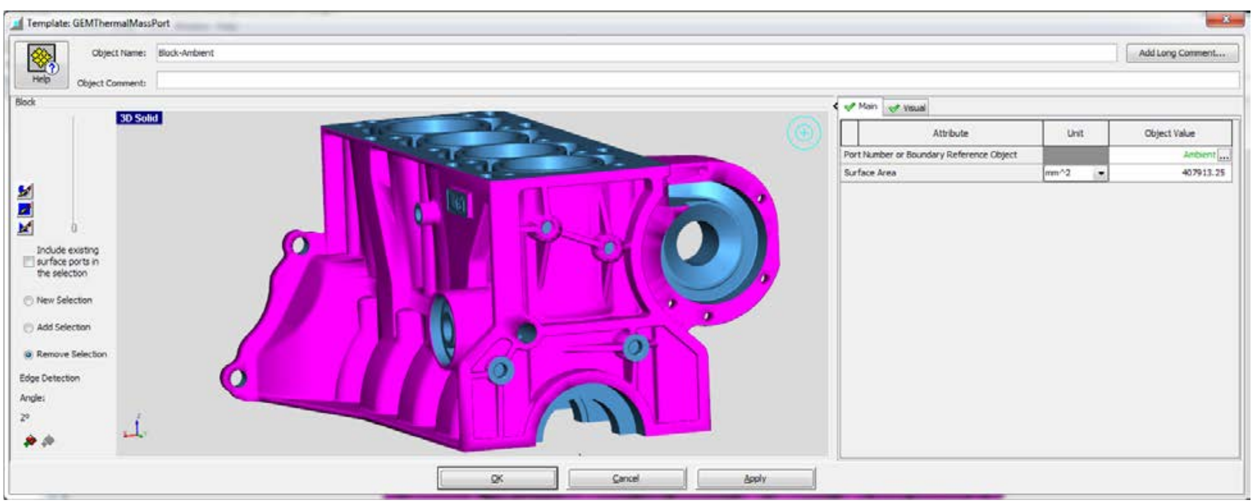
Instead of assigning a Port Number to this connection, a ThermalBound object will be created. When the model is discretized, the ThermalBound will create a Temperature and ConvectionConn linked to the appropriate port on the ThermalMass objects, with the temperature and heat transfer coefficient specified in the ThermalBound. Enter the name "Ambient" into the Port Number or Boundary Reference Object attribute, and double-click on the cell to create the new ThermalBound object. In the Ambient ThermalBound object, create the [ambient-temperature] parameter for the Temperature attribute, and the [ambient-htc] parameter for the Convective Heat Transfer Coefficient.



When the two parameters have been created, click OK to complete the ThermalBound object. Back in the GEMThermalMassPort dialog, name the object Head-Ambient, and then click OK to complete the port.

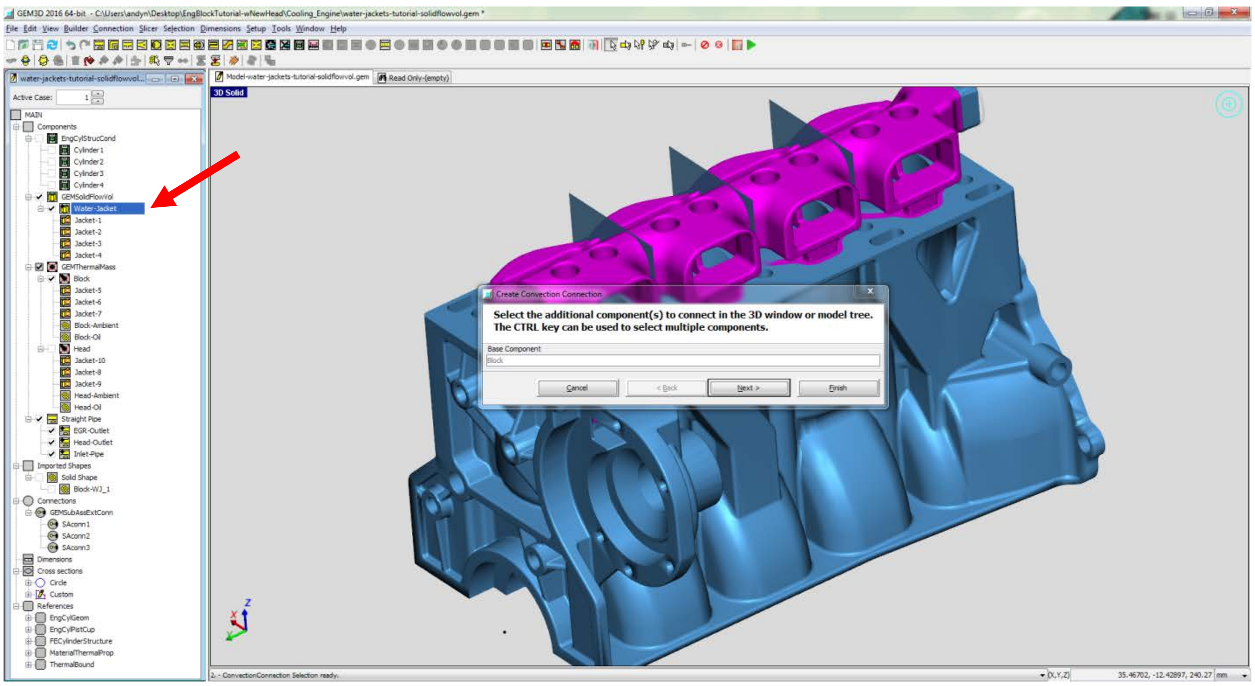


The last port to add is for the ambient connection to the block. Right-click on the Block, and select the Thermal Mass Port command from the menu that appears. Using an edge detection angle of 50 degrees, select the surfaces on the outside of the engine block. The total area selected should be approximately 407913 mm^2. For the Boundary Reference Object, use the Ambient reference object that was previously defined. Name the object Block-Ambient, and click OK to complete the port definition.

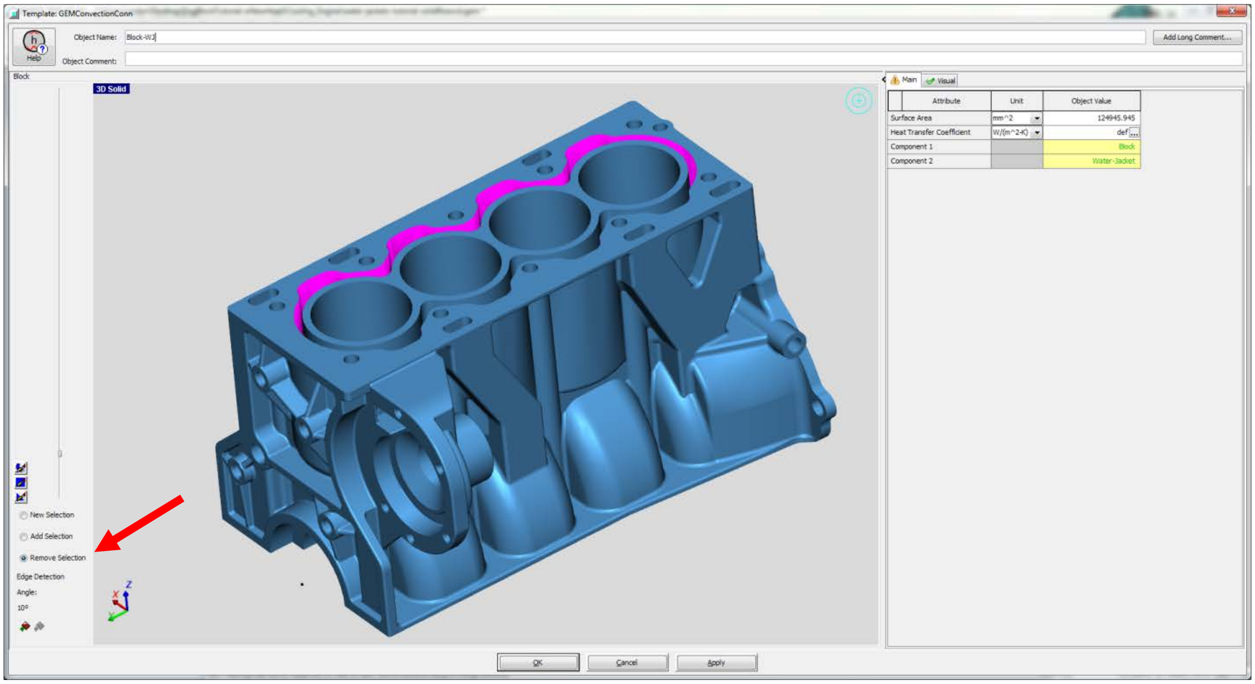


1.4.3 Adding Convection Connection Between Block and Water Jacket

The final connection to be defined in GEM3D is a convection connection between the block thermal mass and the water jackets. To create a new connection, right-click on the block component, or use the Connection menu, to select "Add Convection Connection…" This will bring up a dialog to select the component(s) that will be in contact with the thermal mass. In the model tree on the left, select the "Water-Jacket" GEMSolidFlowVol, and click Next.



In the GEMConvectionConn window that follows, the total contact surface between the Block thermal mass and water jacket is highlighted. Part of this surface must be removed from the selection, since the water jacket flow volume will later be connected to the EngCylStrucCond parts and this surface area must not be double-counted. Using an edge detection angle of 10 degrees, use the "Remove Selection" option and select the portion of the highlighted area on the cylinder liners. Check that the highlighted surface appears correct, and that the calculated surface is approx. 124946 mm^2. Name the connection Block-WJ, and click OK to exit the window, then click Finish to complete the creation.

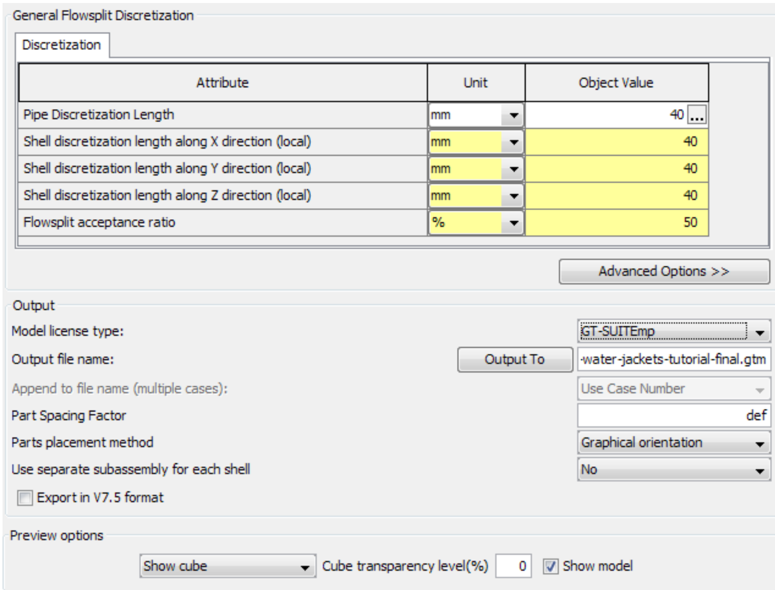


Save the model file when this step is complete.

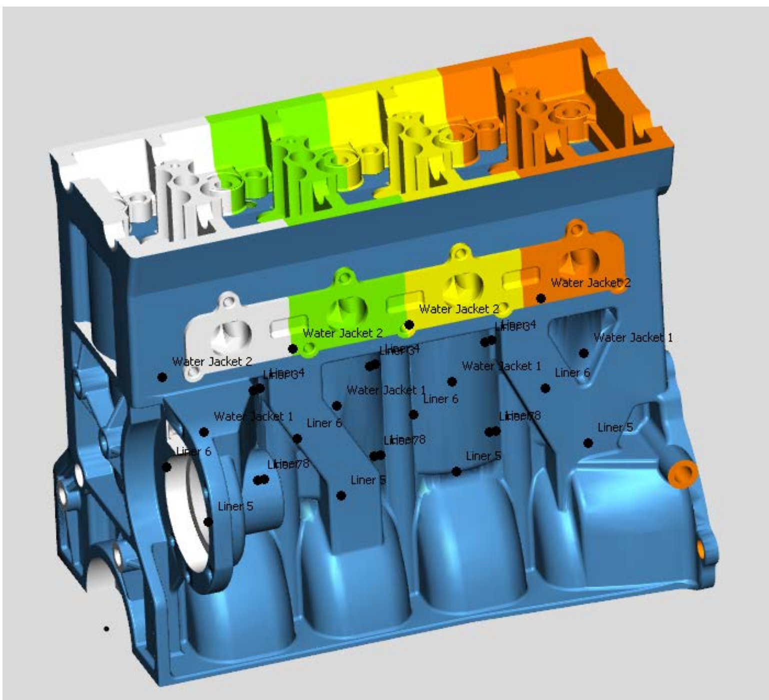
1.5 Discretizing the GEM3D model

For the completed GEM3D model, please see water-jackets-tutorial-final.gem in the tutorial directory.

The next step is to convert the model in GEM3D to GT-ISE pipes, flowsplits, and thermal masses. This process is called discretizing the model, because the shell and thermal mass components in the model will be divided into smaller parts. Open the Discretization window from the File > Export GTM command. The default values for each attribute are OK. Set the Model license type to GT-SUITEmp if it is not already. The Discretization window should appear as shown:



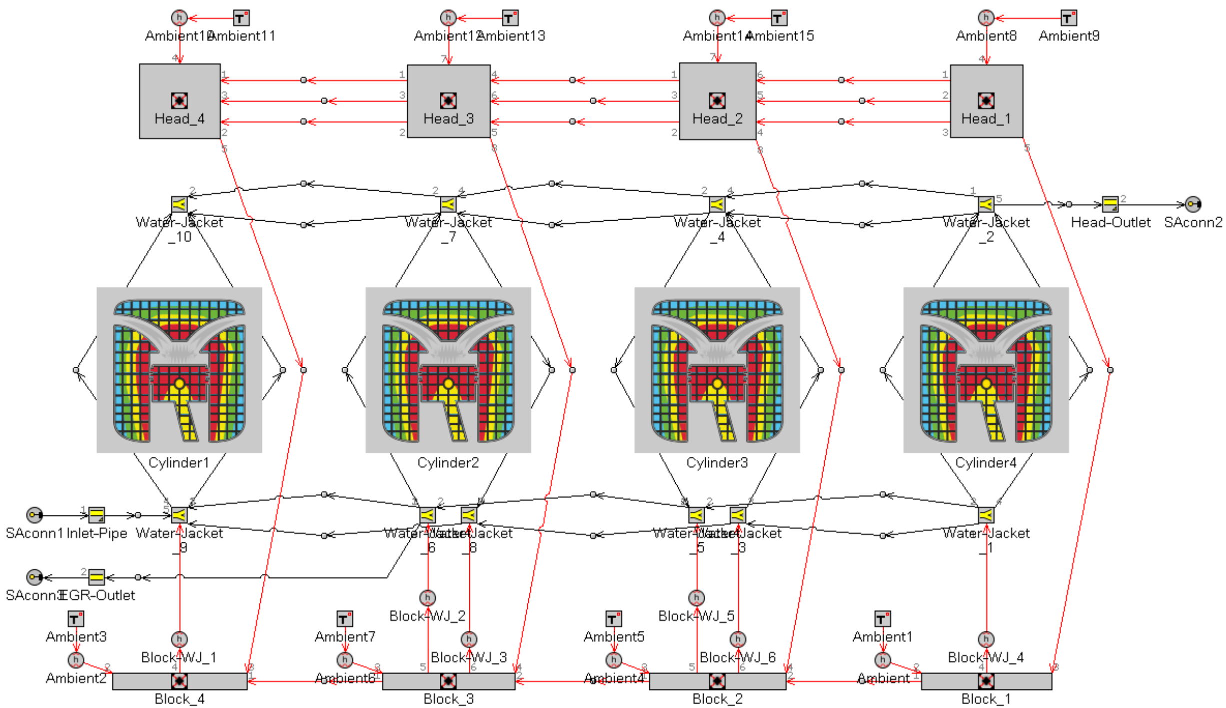
Click the Preview button, and the individual masses and flowsplits will be visible:



The Discretize button will create the .gtm file given in the Output file name attribute. Click this button, and then click Open in GTise on the window that appears when discretization is finished.

1.6 Linking the Model in GT-ISE

After the water-jackets-tutorial.gtm is created, the separate systems will be created on the map. With some organization, the model will appear as shown:



The model was organized such that the head masses and water jackets are at the top section of the map, with the rest of the block down below. Note that the block water jackets for cylinders 2 and 3 have each been divided into 2 flow volumes (because unlike cylinders 1 and 4, the jackets do not wrap continuously around each cylinder, thus creating 2 separate volumes when discretized), and note that the block water jacket volumes are connected to the block thermal masses via the ConvectionConn parts defined in section 1.4.3 of this tutorial. These separate systems must still be correctly linked together to account for the heat transfer between all components. The different types of links will be discussed in the next sections.

1.6.1 Connecting Water Jackets to the EngCylStrucCond

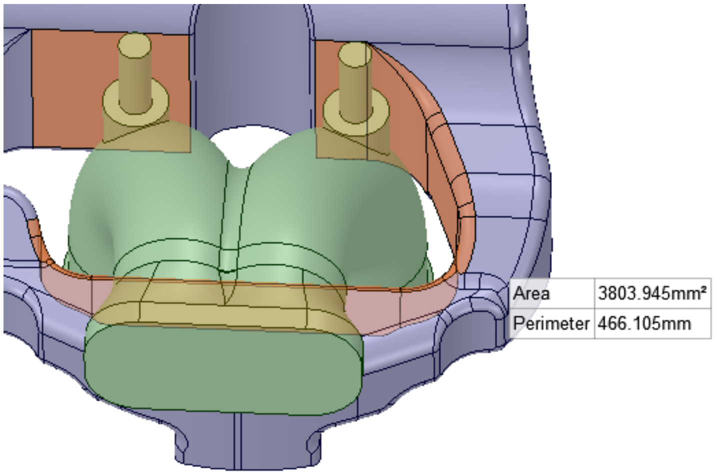
The first connections to make are between the water jacket volumes and the EngCylStrucCond components. These connections will have their surface area imposed based on the water jacket geometry, and their heat transfer coefficients will be imposed from CFD results. The heat transfer coefficients will be discussed further during the calibration section of this tutorial. If necessary, see the Tutorial-WJ-Areas.xlsx file in the tutorial directory for the surface area results.

In GT-SPACECLAIM, open the head water jacket geometry file which was previously saved. If necessary, the water jacket geometry can be re-created from the original head geometry using the Volume Extract operation. Using the Measure tool, calculate the surface areas on the water jacket for each cylinder corresponding to the following locations:

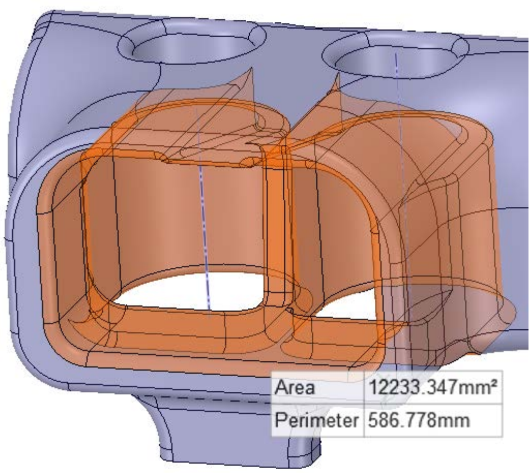
- Exhaust Port
- Intake Port
- Exhaust Guide
- Intake Guide

The measured values can be found in the table below (total surface areas in mm^2):

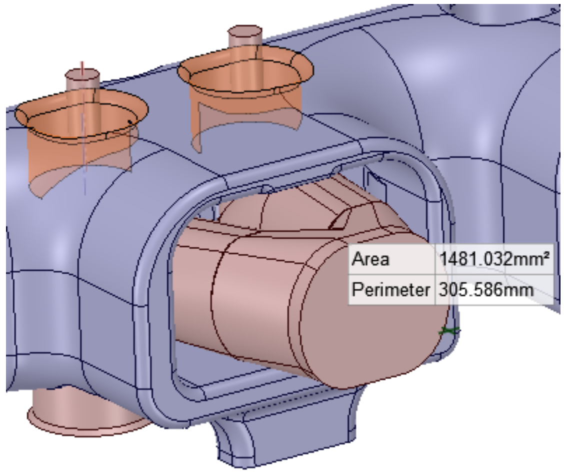
Head Surface	**Cylinder 1**	**Cylinder 2**	**Cylinder 3**	**Cylinder 4**
Intake Ports	3803.95	3803.95	3803.95	4346.32
Intake Guides	0	0	0	0
Exhaust Ports	12233.35	12233.35	12233.35	12233.35
Exhaust Guides	1481.03	1481.03	1481.03	1481.03



Intake Port Area



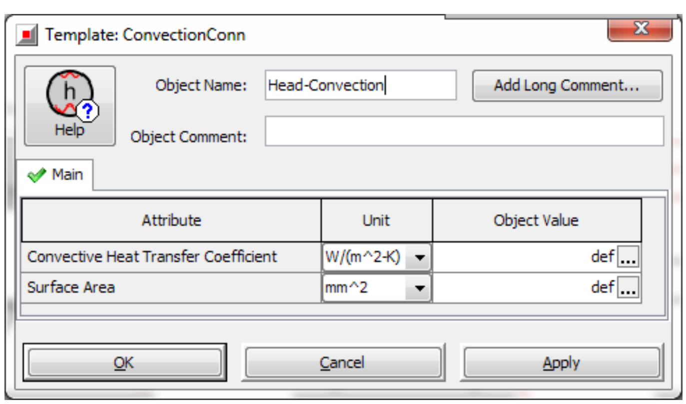
Exhaust Port Area



Exhaust Valve Guide Area

Back in GT-ISE, open the flow split objects that represent the water jacket volumes. In the Thermal folder, ensure that the Wall Temperature from Connected Thermal Primitive option is selected. This will allow the water jacket volumes to be connected to the thermal masses and EngCylStrucCond parts in this model.

Next, create a new 'ConvectionConn' object by double-clicking the template in the model tree, and title it "Head-Convection." The Convective Heat Transfer Coefficient attribute will be left as "def" for now, this will be overwritten during the calibration section of this tutorial. The Surface Area can also be left as "def", part overrides will be used to give the individual surface areas for each connection. Click OK to create the object as shown below.



Because the intake valve guides are not actively cooled in this model (the water jackets are not in contact with the valve guides), six convection connections are needed between the water jacket volume and EngCylStrucCond for each cylinder. Two connections are needed for the intake ports, two for the exhaust ports, and two for the exhaust valve guides. Next to Cylinder 1, create 6 new parts from the Head-Convection object.

The first two parts should be connected from link ID 50 on the EngCylStrucCond (Port 1 Coolant) and link ID 51 (Port 2 Coolant) to the water jacket volume. For the surface area, the areas measured in the above table are for the total area in each cylinder water jacket. For the ConvectionConn parts connected to the intake ports, a part override of 1901.98 mm^2 (= 3803.95/2) should be used for the surface areas.

The next two parts should be connected from link ID 52 on the EngCylStrucCond (Port 3 Coolant) and link ID 53 (Port 4 Coolant) to the water jacket volume. For these ConvectionConn parts, a part override of 6116.68 mm^2 (= 12233.35/2) should be used.

The last two parts should be connected from link ID 46 on the EngCylStrucCond (Valve 3 Guide Coolant) and link ID 47 (Valve 4 Guide Coolant) to the water jacket volume. For these ConvectionConn parts, a part override of 740.52 mm^2 (= 1481.03/2) should be used.

Repeat this operation for the remaining 3 cylinders to complete the connections from the head ports on the EngCylStrucCond to the head water jackets. Note that the surface area should be different on the Cylinder 4 intake ports (2173.16 mm^2 for each port) due to a different measured surface area from the geometry.

The next connections to make are from the EngCylStrucCond to the block water jackets. These surface areas have also been measured from the structure geometry and are shown in the table below (in mm^2):

Block	Cylinder 1	Cylinder 2	Cylinder 3	Cylinder 4
Liner	28744.4	21297.6	21297.6	28744.4

Two connections are needed from each cylinder to the block water jackets. Create a new ConvectionConn object called "Block-Convection", with default values for the heat transfer coefficient and surface area. Then, create two parts on the map for each cylinder; one for link ID 56 (Liner Water Jacket 1) on the EngCylStrucCond and one for link ID 57 (Liner Water Jacket 2). The surface areas given in the table above are a total value for each cylinder. The surface area part override for each ConvectionConn should be half of the total area, because there are two equally sized water jacket ports per cylinder. Also note that the two links from cylinders 1 and 4 are connected to a single fluid volume, while the two links from cylinders 2 and 3 connect to separate volumes.

1.6.2 Connecting Water Jackets to the Structure masses

After the EngCylStrucCond has been connected to the water jackets, the remaining water jacket area must be accounted for by linking and finalizing connections to the external masses representing the rest of the block and head.

For the head, the remaining areas can be calculated by taking the total area of each flowsplit output from GEM3D, and subtracting off the surface area specified in the EngCylStrucCond connections. For the current model, the remaining areas are shown in the table below:

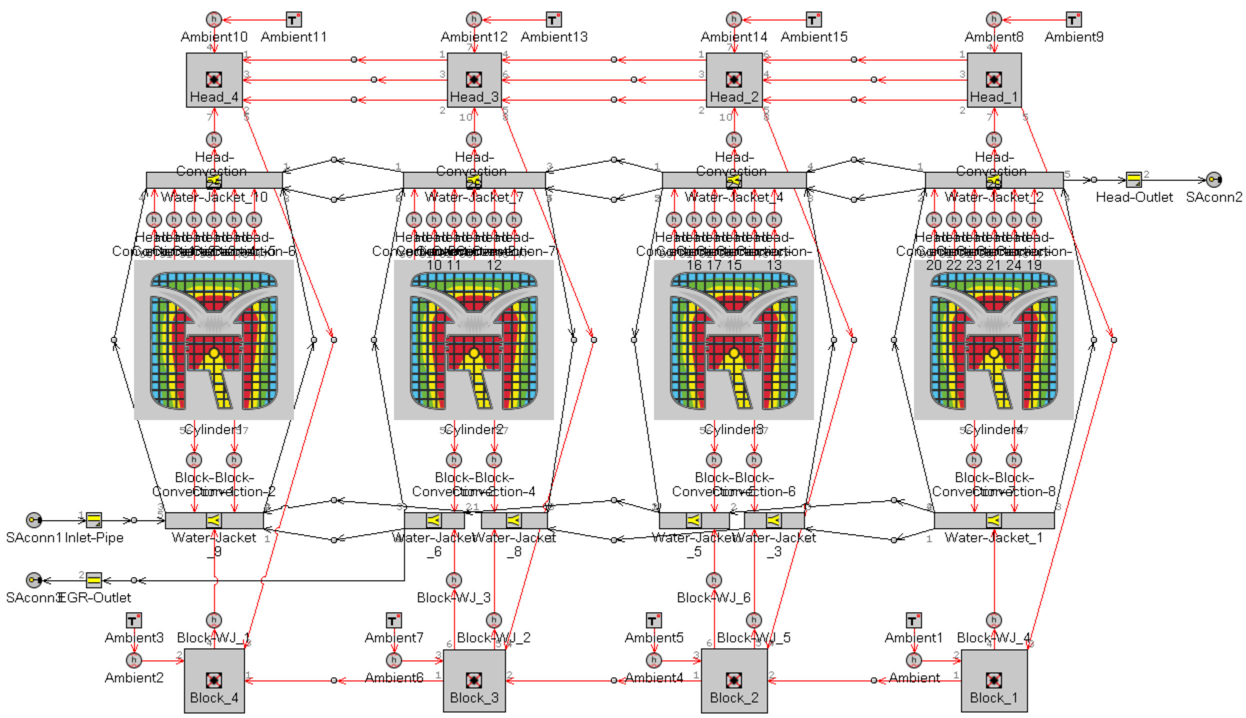
Head Surface	Cylinder 1	Cylinder 2	Cylinder 3	Cylinder 4
Total Flowsplit Area	39342.45	36730.88	36730.83	42436.03
EngCylStrucCond	17518.33	17518.33	17518.33	18060.7
Remaining Area	21824.12	19212.55	19212.55	24375.33

Create one more ConvectionConn part per cylinder from the Head-Convection object. For the surface area part overrides, use the "remaining area" specified in the above table. Before connecting the head thermal masses to the head water jackets through these connections, an additional connection port on the head thermal mass must be created. For each object, create a new port utilizing the following dimensions:

Attribute	**Head 1**	**Head 2**	**Head 3**	**Head 4**
Port Name	WJ	WJ	WJ	WJ
Distance to Mass Center	[head-wj-distance]	[head-wj-distance]	[head-wj-distance]	[head-wj-distance]
Cross Section Area	21824.12	19212.55	19212.55	24375.33
Emissivity	ign	ign	ign	ign

The Distance to Mass Center will be created as a parameter; this value will be adjusted during the calibration process to match the total heat rate from the head to the water jackets and the head temperatures. Once these ports have been added to each thermal mass, connect them to the water jacket through the ConvectionConn.

The block water jacket volumes were automatically connected to the block masses in GEM3D, so these connections and ports do not need to be created here. However, we will create another parameter for the "Distance to Mass Center" attribute, also for adjustment during calibration. Open each "Block" ThermalMass part and edit each port connected to the water jacket volumes to overwrite the calculated distance to mass center with a new parameter [block-wj-distance]. This value will be adjusted during the calibration process to match the total heat rate from the block to the water jackets and the block temperatures. The port names may also be changed to "WJ" if desired. When this has been completed, the water jacket flowsplits can be re-sized to make the connections easier to follow. The model should resemble the picture:



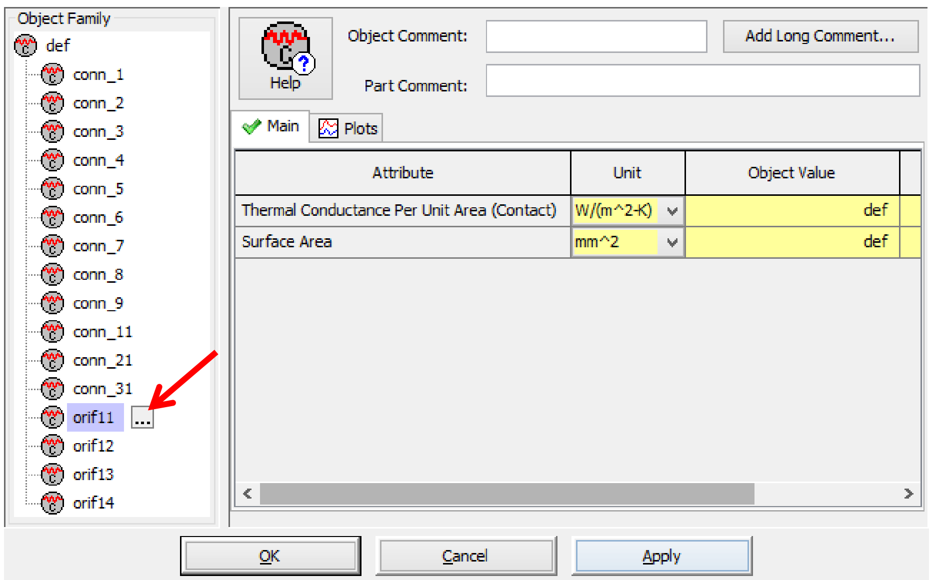
The thermal connections for the water jackets are complete. For the remaining pipes, the wall should be adiabatic; in the Inlet-Pipe, EGR-Outlet, and the Head-Outlet pipes, change the Wall Temperature Method in the Thermal folder to the Adiabatic option as shown below:

	Attribute	Unit	Object Value
	Wall Temperature Method		
○	Imposed Wall Temperature	See Cas...	[WallTemp]
○	Calculated Wall Temperature		
	Wall Temperature Solver Object		ign
	Initial Wall Temperature	See Cas...	[WallTemp]
○	Wall Temperature from Connected Thermal Primitive		
◉	Adiabatic		

1.6.3 Connecting the Block to the Head

The connections between the block thermal masses and the head thermal masses were created during the GEM3D discretization process. However, the default connections need to be modified to account for the conduction through the head gasket. Create a new ConductanceConn object, named Head-

Gasket. The default Surface Area is sufficient, but an equation will be used for the Thermal Conductance attribute; enter "=1/[hg-resist]" for its value. Select units of m^2-K/W for the [hg-resist] parameter (in the "Thermal Resistance (Unit Area)" units category. must use units will be used during the calibration process. On the map, open up each of the connections between the Head mass and Block mass. Next to the part name in the Object Family tree, choose the Value Selector button to change the parent object from a "def" ConductanceConn to the Head-Gasket ConductanceConn. The parent object can also be changed by right-clicking on the ConductanceConn icon next to the part name.



When all of the parent objects have been changed, the ConductanceConn icon should appear for the connections on the map, instead of the small default "bullet" icon.

1.6.4 Connecting the EngCylStrucCond to the Structure

The next thermal connections to make to the model are between the EngCylStrucCond and the thermal masses. These connections will allow for heat transfer directly to the thermal mass parts from the engine structure, to represent the areas on the outside of the cylinder structure that are not covered by the water jackets. The first connections are to the block from the lower portions of the EngCylStrucCond. Create new ports on the "Block" parts using the following attributes.

Attribute	Block 1	Block 2	Block 3	Block 4
Port Name	Liner	Liner	Liner	Liner
Distance to Mass Center	[block-structure-distance]	[block-structure-distance]	[block-structure-distance]	[block-structure-distance]
Cross Section Area	9650.97	9650.97	9650.97	9650.97
Emissivity	ign	ign	ign	ign

The [block-structure-distance] parameter will be used for calibrating the model to adjust the resistance between the block mass and the EngCylStrucCond. The Cross Section Area can be measured in SpaceClaim, and is determined from the cylinder geometry, by calculating the area on the outside of the cylinder between the bottom of the water jacket and the bottom of the cylinder.

Next, create a new ConductanceConn object called Structure-Cylinder. For the Conductance attribute, use a value of 50000 W/m^2-K. This value is generally used in place of the default value when connecting to an EngCylStrucCond. Using the default value when connecting a ThermalMass or ThermalNode to an EngCylStrucCond (a FE surface) will create a thermal circuit which "short-circuits" the nodes on the EngCylStrucCond. The resistance path through the ConductanceConn will be lower than the resistance between nodes on the structure, so the outside of the structure would be isothermal. The recommended lower conductance value creates a greater resistance than what occurs among the structure nodes, so temperature variations are maintained.

The new ports on the block thermal masses will be connected to the following EngCylStrucCond ports:

Cylinder	Cylinder 1	Cylinder 2	Cylinder 3	Cylinder 4
Link ID Number	63, 64, 66	63, 64, 65, 66	63, 64, 65, 66	63, 64, 65

For the connections between the EngCylStrucCond and the head, create new ports on the head thermal masses using the following attributes:

Attribute	**Block 1**	**Block 2**	**Block 3**	**Block 4**
Port Name	Head	Head	Head	Head
Distance to Mass Center	[head-structure-distance]	[head-structure-distance]	[head-structure-distance]	[head-structure-distance]
Cross Section Area	3617.9	3617.9	3617.9	3617.9
Emissivity	ign	ign	ign	Ign

The [head-structure-distance] parameter will be used for calibrating the model to adjust the resistance between the head mass and the EngCylStrucCond. The Cross Section Area is determined from the cylinder geometry, calculating the total area on the top of the cylinder and subtracting out the port opening area. Using the parts created from the Structure-Cylinder ConductanceConn object, connect the new ports on the head masses to port 43 (the Head Coolant link) on the EngCylStrucCond. This will allow the cylinder structure to conduct heat directly to the head masses, in addition to the heat already going to the coolant.

1.6.5 Creating Oil Volumes in the Block and Head

Another important factor in the engine block thermal system is the heat exchange between the engine oil and the block. Two additional volumes will be created to represent the oil pooling in the head before returning to the oil pan, and the oil splashing in the crankcase. These volumes will be connected to the head thermal masses, the block thermal masses, and two ports on the EngCylStrucCond representing the underside of the piston and cylinder walls.

To obtain the volume for these oil components, the previously measured surface areas from GEM3D can be used. In Section 1.4.2, the oil ports were created by selecting the surfaces on the block and head that were in contact with the oil. The measured surface areas were: 23273 mm^2 for the head, and 137295 mm^2 for the block. A 1 mm film thickness is assumed on these surfaces, so create two new FlowSplitGeneral objects with the following attributes:

Name	**Oil-Head**	**Oil-Block**
Volume (mm^3)	23273	137295
Surface Area (mm^2)	23273	137295
Initial State Name	Oil-Initial	Oil-Initial
Wall Temperature Method	Wall Temperature From Connected Thermal Primitives	Wall Temperature From Connected Thermal Primitives
Boundary Angles (deg)	0, 180 (2 ports)	0, 180 (2 ports)
Characteristic Length (mm)	50	50
Expansion Diameter (mm)	50	50

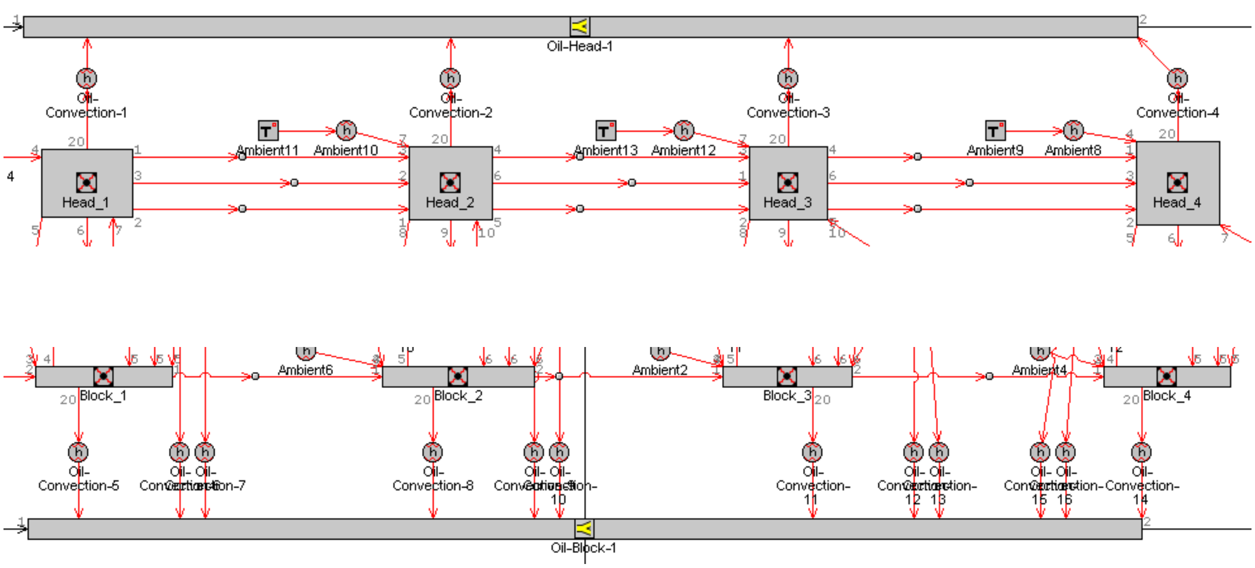
The attributes for the Oil-Initial conditions using the FluidInitialState template are:

Pressure	[oil-pressure]
Temperature	[oil-temperature]
Composition	Oil-5W30-62-10cSt

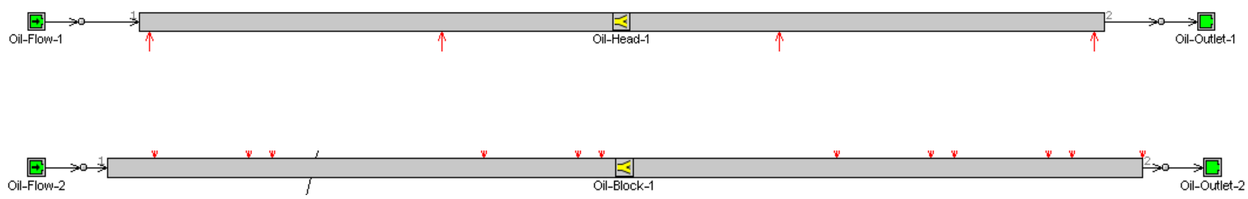
Ensure that in the Thermal folder, the Wall Temperature from Connected Thermal Primitives option is selected. Next, review the ports that were created in the head for the oil. The Link ID number/Port name in each part was 20 (so that it could be easily found during the connecting process); this port may be renamed "Oil" if desired. GEM3D also calculated the distance to mass center and cross sectional area for each individual part from the original marked surface. Because the heat rate to the oil will be calibrated to match the reference conditions, a parameter should be used for the Distance to Mass Center value for this port. The [head-oil-distance] parameter will adjust the resistance between the head mass and the oil volume. Add this parameter to the oil boundary for each of the head masses.

For the block, the [block-oil-distance] parameter will be used for calibrating the model to adjust the resistance between the block mass and the oil volume. Add this parameter to the oil boundary for each of the block masses.

To make the connections, create a new ConvectionConn object called Oil-Convection. For the Heat Transfer Coefficient, use a value of 400 W/m^2-K. This value is typically used to represent the oil splashing in the crankcase and the pooling in the head. The heat transfer coefficient can be adjusted during the calibration process if necessary. Create 16 new parts from the Oil-Convection object; one between each head thermal mass and the head oil volume, one between each block thermal mass and the block oil volume, and two between each EngCylStrucCond part and the block oil volume. On the thermal masses, the oil should be connected to port 20 (or "Oil" if the port was renamed). On the EngCylStrucCond, the oil should be connected to ports 42 and 58. The completed model should appear as shown:



In addition to the oil volumes, boundary conditions need to be created for the oil flow for calibration purposes. From the template library, create a new EndFlowInlet object called Oil-Flow. Select the Volumetric Flow Rate radio button, and enter a value of 1 L/s. For the temperature, use the parameter [oil-temperature] which was previously used for the oil initial conditions, and select ‘Oil-5W30-62-10cSt’ for the Composition. Create two Oil-Flow parts on the map from this object, next to the previously created oil volumes. Next, create a new EndEnvironment object called Oil-Outlet. For the Pressure (Absolute), use the [oil-pressure] parameter. Use the [oil-temperature] parameter for the Temperature attribute, and select ‘Oil-5W30-62-10cSt’ for the composition. Place two Oil-Outlet parts on the map on the other side of the oil volumes from the EndFlowInlet parts. When finished, the oil flow paths should appear as shown:



1.7 Calibrating the Model

The calibration of engine thermal models typically requires CFD support. Because of the inherently 3-D nature of flow through the water jackets, the heat transfer correlations used for the 1-D flowsplit generally under predict the heat transfer coefficients. Furthermore, the reduced number of volumes and connections utilized in this model means that the pressure losses through the water jackets will also require some calibration. The simplifications made for 1-D conduction in the head and block will also require adjustment to match the metal temperatures.

1.7.1 Required data for calibration

If a 3-D CFD model is available, a conjugate heat transfer analysis should be performed for at least two different steady state flow and thermal conditions. The goals of the fully detailed model would be to provide:

- Pressure loss vs. flow rate through the water jackets from the inlet to the various outlets.
- Heat transfer coefficients vs inlet flow rate for the water jackets. These coefficients could either be an area averaged value for each flow volume, or the averaging could be performed for the block and head. The heat transfer coefficients in the block and in the head are typically different enough that two separate relationships vs inlet flow rate are recommended.
- Heat transfer rate to the coolant, oil, and environment at each model condition.
- Peak metal temperatures for the block and head at each model condition. A more detailed temperature distribution would allow for a better match of the thermal mass temperatures to the average metal temperatures.

If a conjugate heat transfer analysis is not available, the pressure loss, heat transfer rate, and metal temperatures can be obtained from tests run on an engine dynamometer.

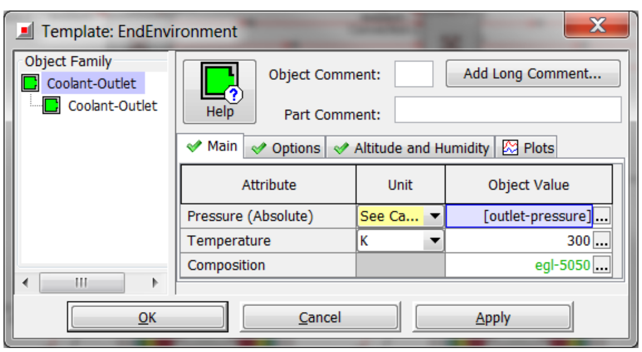
For this case, sample results have been provided in the Boundary-Conditions.xlsx spreadsheet in the tutorial directory.

1.7.2 Calibrating the model pressure loss

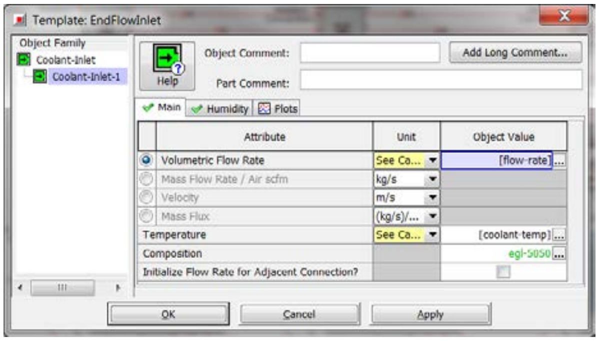
For engine heat distribution models discretized with this level of detail, the pressure losses calculated by GT-SUITE will not be the same as the measured ones; calibration is needed. The idea behind the modeling approach is to impose the pressure losses with a PressureLossConn at each outlet of the engine and to keep the losses calculated inside the flowsplits (friction multiplier set to zero) and connections minimal. Therefore create two new PressureLossConn objects in the model. One should be named Head-PDrop and will be placed at the outlet of the head water jackets, downstream of the Head-Outlet pipe. When placing this part, remove the SAconn2 part that is currently downstream of the Head-Outlet pipe.

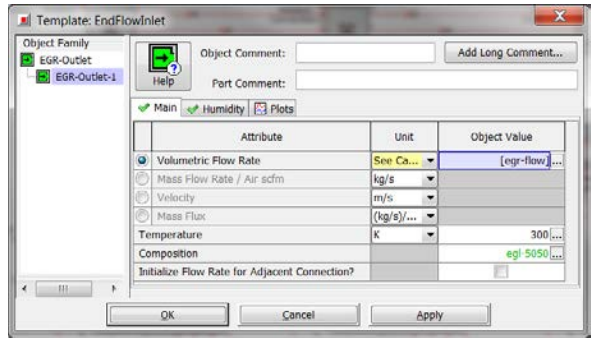
Another should be named EGR-PDrop and will be placed upstream of the EGR-Outlet pipe. When placing this part, remove the SAconn3 part that is currently downstream of the EGR-Outlet pipe. In each of these PressureLossConn objects, use a FlowPDropTableRef reference object to define the pressure loss. This reference object will fit a pressure loss coefficient vs Reynolds number curve for each entered data set. Use the provided "Pressure Loss" data sets in Boundary-Conditions.xlsx for each pressure loss object.

In addition to creating the pressure loss objects, the boundary conditions for the fluid flow should also be created. In the Boundary-Conditions.xlsx, under Flow Conditions, the engine inlet coolant flow rate, the EGR outlet flow rate, as well as the cylinder head outlet pressure are given for two engine operating points. This information will now be used to impose the boundary conditions to the model. Therefore create an EndEnvironment at the outlet of the head water jacket (to impose the pressure), with the following attributes:

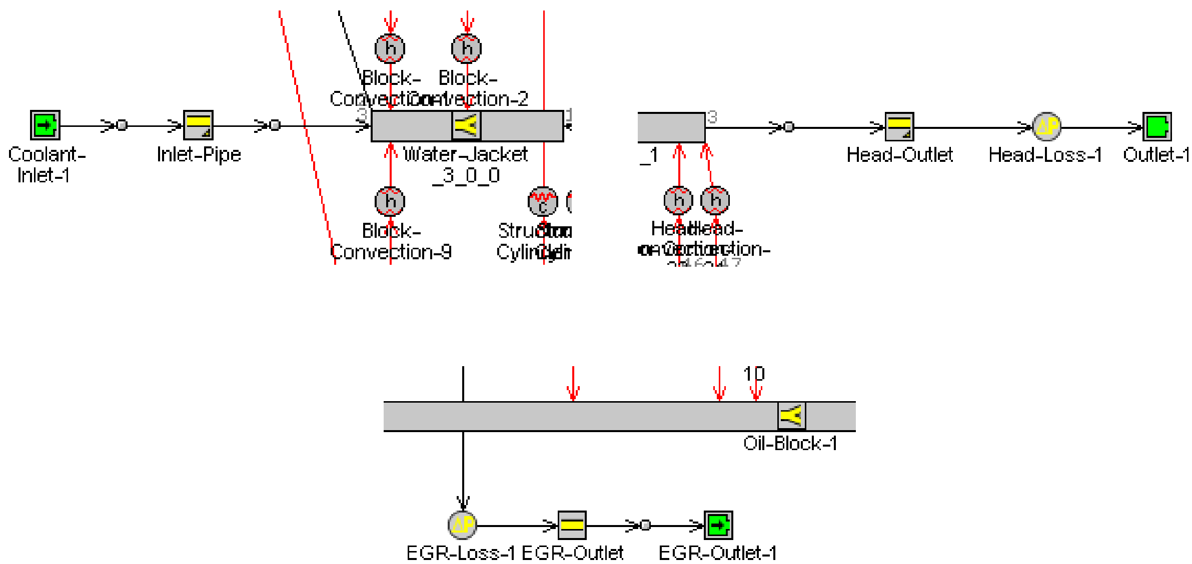


To impose the flow rates at the water jacket inlet and the EGR outlet, use EndFlowInlet objects with the following attributes:

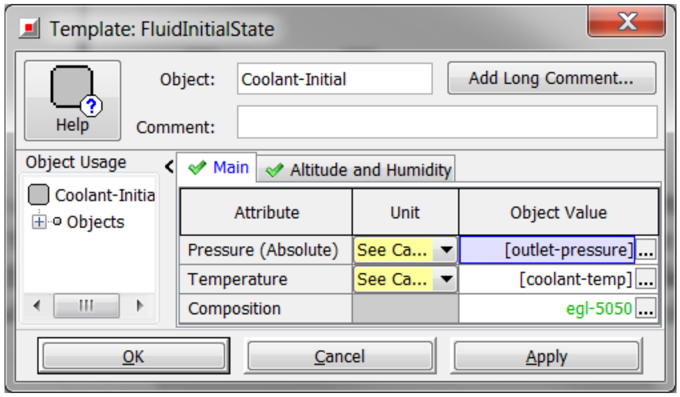




Place the Coolant-Inlet upstream of the Inlet-Pipe, and the EGR-Outlet downstream of the EGR-Outlet Pipe. When completed, the model should appear as shown:



Once the boundary conditions are connected, the initial conditions for the water jackets should be defined. In one of the coolant pipes or water jacket flowsplits, select the reference object named in the Initial State Name attribute. This name should have been filled out from GEM3D (Coolant-Initial). If not, it will need to be manually added to the rest of the coolant system. Create a new FluidInitialState reference object with the following attributes:



The attribute values for the created parameters will be filled out later on in CaseSetup. This will be discussed in section 1.7.5.

In this section the approach to impose the coolant pressure drop due to the engine based on multiple measured data points was discussed. In case there are fewer points available, for example only two, the approach to use the FlowPDropLossCoef inside the PressureLossConn object (instead of the FlowPDropTableRef) is recommended. It is convenient to use the built-in Direct Optimizer to vary the Pressure Loss Coefficient in order to match the target measured points. Please consult the Direct Optimizer help for further reference.

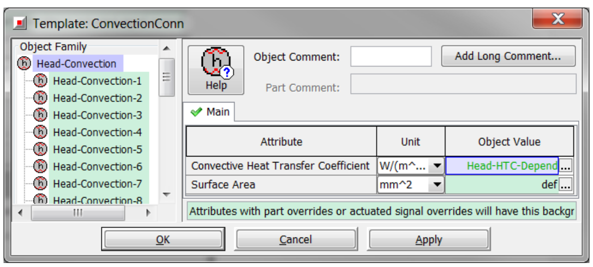
1.7.3 Imposing the heat transfer coefficients

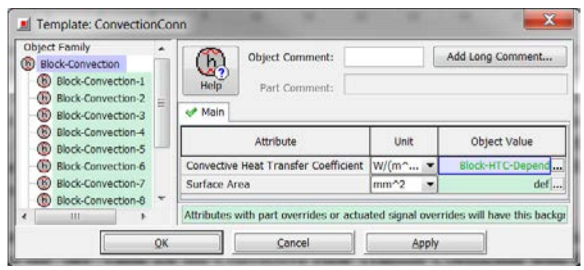
As mentioned previously, the heat transfer coefficients in the head and block water jackets are typically under predicted by the default correlations used in GT-SUITE. If detailed coefficients are known, they can be imposed directly in the convection connections using a dependence on the block inlet flow rate. If the coefficients are not known but test results are available, the overall heat transfer rate can be calibrated using a heat transfer multiplier in the water jacket volumes.

For this model, the averaged heat transfer coefficients for various flow rates are given in Boundary-Conditions.xlsx. To implement these in the model, create a new RLTDependenceXY object from the template library, called Head-HTC-Depend. For the Input RLT Variable attribute, select the Average Volume Flow Rate (Inlet) in [L/s] from the Inlet-Pipe part. The Initialization for First RLT Period attribute specifies the initial value for the RLTDependenceXY, used through the first RLT update interval. For this attribute, select the Initial Y Output option and enter a value of 5000 W/m^2-K.

For the Dependence Object, create a new object named Head-HTC. Double-click on the reference object to define it, and create a new XYTable. Copy the flow rate and heat transfer coefficient information into the table from the spreadsheet, using the flow rate for the X values and the heat transfer coefficient Y values. The flow rate is already available in the unit of [L/s], therefore no further adjustment of data is needed. When the table has been filled in, click OK in the XYTable to create the reference object and again in the RLTDependenceXY. Repeat these steps for the Block-HTC-Depend reference object and Block-HTC table to create the block heat transfer coefficient reference, using the values given in the spreadsheet.

When both reference objects have been created, open the Block-Convection and Head-Convection ConvectionConn objects, as well as all Block-WJ ConvectionConn objects that were created by GEM3D. Replace the 'def' value for the Convective Heat Transfer Coefficient with the respective RLTDependenceXY objects. Because the change for the heat transfer coefficient was made at the object level, all of the dependent parts on the map will be automatically updated to use the new function.

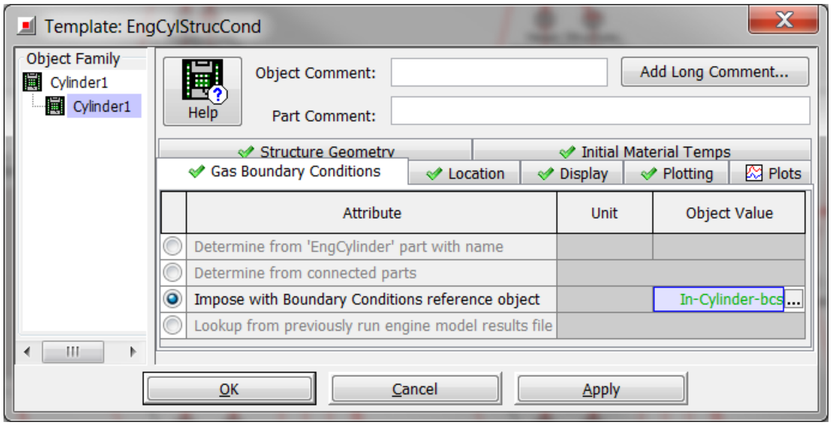




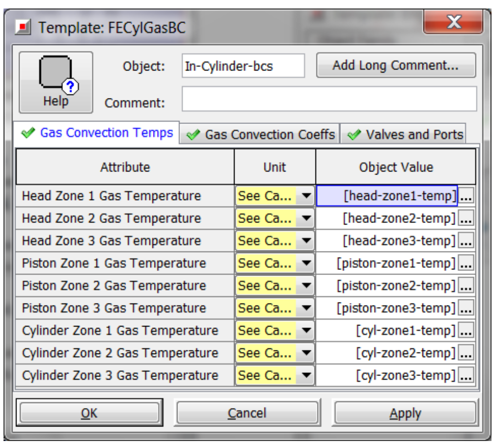
1.7.4 Imposing the engine boundary conditions

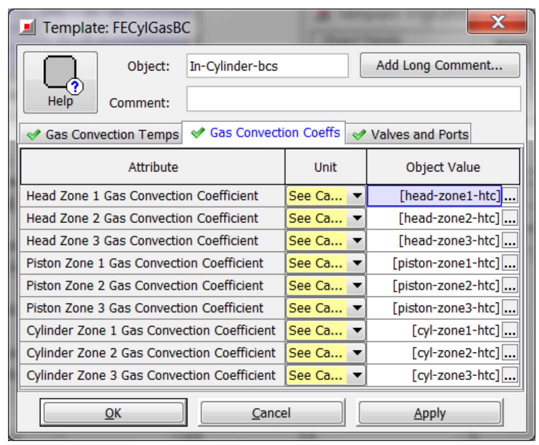
For the in-cylinder boundary conditions, values applied to the thermal model should match the values from the 3D CFD simulation or test data in order to calibrate and validate the model. In this case, results from a GT-POWER simulation will be used. The Diesel_Engine_Tutorial.gtm file in the tutorial directory is the detailed engine model which corresponds to the actual engine geometry in question. There are multiple ways to link a GT-POWER simulation to the current thermal model. A direct connection can be made between the engine cylinders and the EngCylStrucCond when running the model simultaneously. For a stand-alone thermal model of a transient simulation, the boundary conditions can be imported from the GT-POWER results based on the speed and load from an EngineState object. The in-cylinder boundary conditions can also be imposed manually using the FECylGasBC reference object. This last approach is typically used for calibrating thermal models due to the direct control and will be used in this tutorial.

Open the EngCylStrucCond template named Cylinder1, and select the 'Impose with Boundary Conditions reference object' in the Gas Boundary Condition Tab. Create a new reference object named In-Cylinder-bcs:

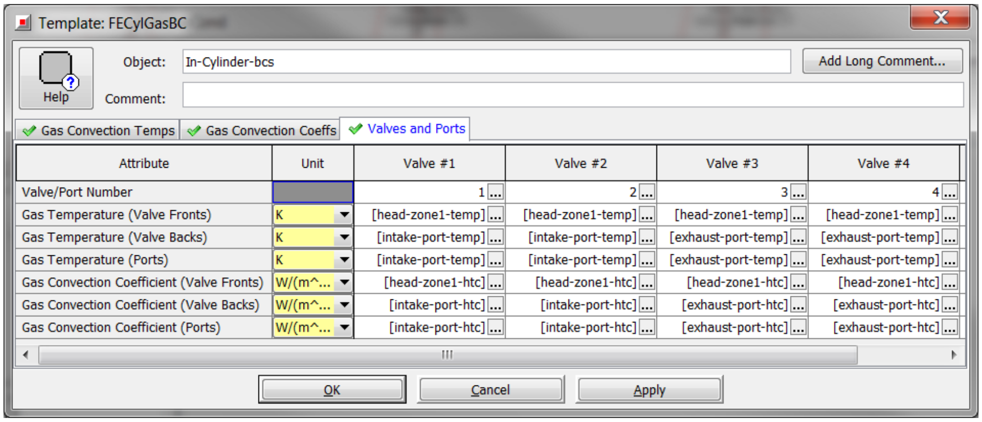


For the Gas Convection Temps and Coeffs folder, parameters will be used for each zone:





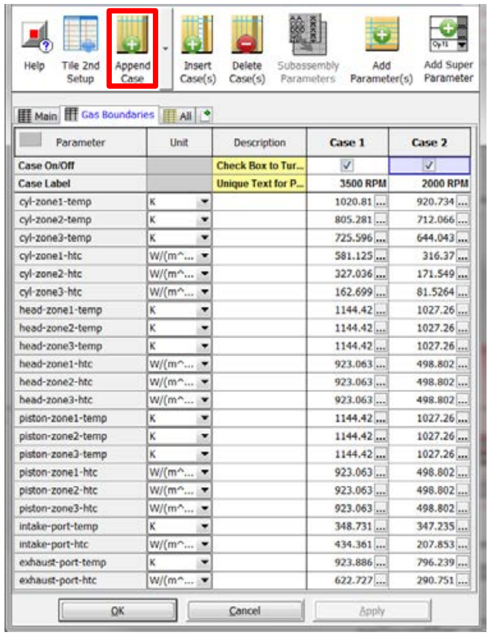
The valves and ports boundaries should be defined using new parameters for the intake and exhaust port temperatures and heat transfer coefficients. The valve radiation temperatures and coefficients are not shown in the diagram below, they should be set to the same values (1000 K, 0 W/m^2-K) as the other radiation attributes:



Once the "In-Cylinder-bcs" reference object is complete, assign it for the imposed gas boundary for all of the remaining 3 Cylinders.

The values for the in-cylinder boundary conditions are given in the Boundary-Conditions.xlsx spreadsheet. These boundary conditions were taken from the completed GT-POWER simulation at the specified speed and load points. Open up Case Setup from the Home menu, and add a case to the model using the Append Case button, highlighted in the picture below.

To simplify Case Setup, the gas boundary parameters can be moved to a new folder. To create new folder, use the button at the top of the window. Parameters can be added to the new folder by right-clicking on a parameter and selecting the Move Parameter to Folder command, or by left-clicking and dragging the parameter to the folder's icon at the top of the window. Sort the gas boundary parameters into the new folder, and then copy the boundary conditions in from the spreadsheet. The result looks like this:



1.7.5 Model Setup

The additional boundary conditions for the model should also be defined using the parameters in case setup. The values for the coolant flow rates and temperatures can be entered directly. Note that the EGR flow rate is defined as a negative value in case setup. This is due to the link direction on the map and the convention for the EndFlowInlet. The ambient temperature, heat transfer coefficient, as well as the oil boundaries can also be defined from the spreadsheet. The WallTemp and structure-temp parameters should be set to the coolant temperature. For the distance parameters, these should be set to some initial value (for example: 40 mm). The distance parameters will be changed to calibrate the conduction paths through the model, as discussed in the next section.

Parameter	Unit	Description	Case 1	Case 2
Case On/Off		Check Box to Turn Case On	☑	☑
Case Label		Unique Text for Plot Legends	3500 RPM	2000 RPM
WallTemp	K		358	358
structure-temp	K	Structure initial temperature	358	358
head-wj-distance	mm	Distance to Mass Center	10	10
block-wj-distance	mm	Distance to Mass Center	25	25
block-structure-distance	mm	Distance to Mass Center	20	20
head-structure-distance	mm	Distance to Mass Center	15	15
head-oil-distance	mm	Distance to Mass Center	15	15
block-oil-distance	mm	Distance to Mass Center	30	30
hg-resist	(m^2-K...	Head Gasket Contact Resistance	1.45e-4	1.45e-4
oil-temperature	K	Temperature	398	398
oil-pressure	bar	Pressure (Absolute)	1	1
coolant-temp	K	Temperature	358	358
outlet-pressure	bar	Pressure (Absolute)	1.4	1.27
flow-rate	L/s	Volumetric Flow Rate	2.781	1.567
egr-flow	L/s	Volumetric Flow Rate	-0.278	-0.157
ambient-temperature	K	Temperature	300	300
ambient-htc	W/(m^...	Convective Heat Transfer Coeff...	50	50

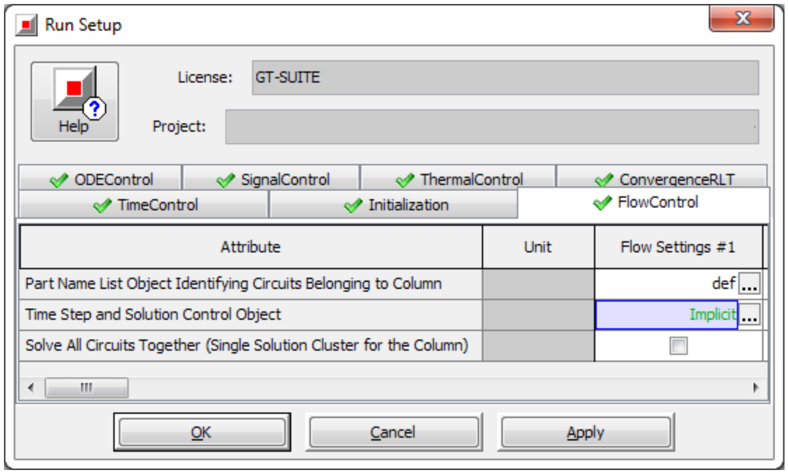
OK Cancel Apply

The next step is to add some information to Run Setup (available in the Home menu) to define the model run time and solution method. In the TimeControl folder of Run Setup, set the Time Control Flag to Continuous and the Maximum Simulation Duration (Time) attribute to 30 seconds. The folder should appear as shown below:

Attribute	Unit	Object Value
Time Control Flag		continuous
Maximum Simulation Duration (Cycles)		
Minimum Simulation Duration (Cycles)		ign
Maximum Simulation Duration (Time)	s	30
Minimum Simulation Duration (Time)	s	ign
Automatic Shut-Off When Steady-State		on
Main Driver (Defines Periodic Frequency)		
Automatic		
Part Name		
Reference Object		
Improved Solution Sequence for Multi-Ci...		☐

OK Cancel Apply

The next area to change is the FlowControl folder. Enter def for the Part Name List... attribute, and use the value selector button to select the "Implicit" reference object from the GT-SUITE Library. The completed folder should appear as shown:



The next folder in Run Setup is the ODEControl folder. Use the Value Selector to import the default ODEControlExplicit object called "Explicit-def" for the Time Step and Solution Control attribute. Even though the name appears in the attribute already, the object must be imported because the model was originally created in GEM3D. When the run settings have been changed, and the objects have been imported, select OK to complete the Run Setup.

More information regarding continuous time flag, implicit solver and steady state criterion can be found in the Flow.pdf manual under File -> Manuals -> Modeling_Theory.

The next settings to change are in Output Setup (available on the Home menu). In the ScoreboardRLTs, the Value Selector should be used to import the default scoreboards from the template library. Even though the names appear in the attributes already, the objects must be imported because the model was originally created in GEM3D. Once the default Scoreboards are imported, the Output Setup window can be closed.

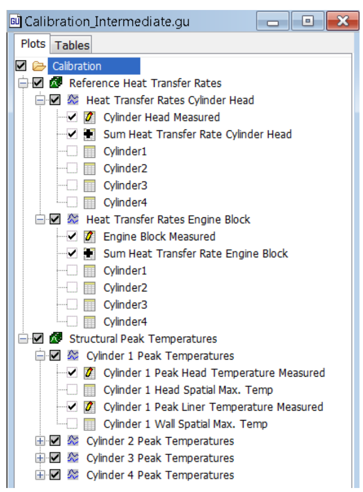
1.7.6 Calibration Guidelines

The goals for the model calibration are, for a given set of boundary conditions, to match the heat distribution to the fluids (if not directly measured, implicitly this information is given if coolant and oil inlet and outlet T, p and flow rate is available) and to match the maximum temperatures in the liner and head.

The heat transfer coefficients are known from CFD and imposed in the model, while heat transfer areas are determined from the real geometry through the previous steps. Because these values are known, the recommended approach is to calibrate the conductive restrictions through the structure in order to get the correct heat rates and fluid-to-wall temperature differences.

The recommended calibration levers are the "Distance to Mass Center" parameters that were created in the Thermal Mass parts. Typically, some initial values are used for these distances roughly based on the model dimensions, and then multiple iterations of the model are simulated to match to the calibration targets. In addition to the conduction distances, other parameters are typically adjusted to match the heat rates and in-cylinder temperatures; the resistance of the head gasket is one such parameter. This value is used in the connections between the block and head ThermalMass parts, and should also be used for the "Head Gasket Contact Resistance" attribute in the FECylinderStructure objects. The heat conducted from the valves to the head can be adjusted by changing the Valve to Seat HTR Coefficient. Another parameter is available if the cylinder inlay is represented, the inlay contact resistance can be adjusted to control the peak cylinder temperature. If the pistons were included in the CFD analysis, or if the calibration targets are taken from test data, the heat transfer coefficient between the piston and the wall can be adjusted using the Skirt to Cylinder, Ring to Piston, and Ring to Cylinder coefficients.

The usage of a Report File (.gu) to track the impact of the parameter variations on the results is highly recommended. Open the supplied *Calibration.gu* from the tutorial directory in GT-POST.



There are two Groups, Reference Heat Transfer Rates and Structural Peak Temperatures. The first one contains two plots, the heat transfer rates for cylinder head and for engine block. In each plot there are explicit datasets representing the measured data, which can also be found in the Boundary-Conditions.xlsx. With the help of a mathematical equation, the sum of heat transfer of each single cylinder head flow volume is created. The links to the results are implicit links pointing to the parts in the water-jackets-tutorial.gtm, therefore it is important to make sure the part names match the ones in the final model and also the file name should be kept as water-jackets-tutorial.gtm. Once the model runs, the CaseRLTs are automatically stored in the report file and can be directly compared to the measured data. This makes it easier to calibrate the model because the relevant targets can be reviewed automatically. The plots and datasets in the second group inside the Calibration.gu are setup in the same way. The RLTs of interest are the Net Heat Transfer Rate RLT for each fluid volume for the heat transfer rates, and the Cylinder Wall/Inlay Spatial Max. Temp and the Head Spatial Max Temp RLTs for the peak temperatures.

To achieve the target values, the following parameters are used in the final model. Note that a parameter was added for the head gasket resistance, used in both the EngCylStrucCond attribute and in the Head-Gasket connections (using the formula =1/[hg-resist]). Because the calibration parameters are not a function of the speed or flow rate, the same values should be used for both cases.

head-wj-distance	mm	Distance to Mass Center	10	10
block-wj-distance	mm	Distance to Mass Center	25	25
block-structure-distance	mm	Distance to Mass Center	20	20
head-structure-distance	mm	Distance to Mass Center	15	15
head-oil-distance	mm	Distance to Mass Center	15	15
block-oil-distance	mm	Distance to Mass Center	30	30
hg-resist	(m^2-K...	Head Gasket Contact Resistance	1.45e-4	1.45e-4

1.7.7 Preparing the Model for Transient Operation

So far, the tutorial has been focused on the steady state operation of the model. Before the thermal model can be run in a transient simulation, an important change needs to be made. The thermal masses in the model were created from the complete CAD model, as were the EngCylStrucCond components, containing a small disc of the cylinder head, the valve ports as well as the cylinder liner. This means that the mass of the mentioned components are included twice in the model, in the discretized mass and in the finite element components. At steady state, the thermal capacitance of material is ignored, but this additional mass will affect the warm-up of the model in a transient simulation.

To correct this, the appropriate finite element mass should be subtracted from each thermal mass. This mass can be obtained in GT-POST from the EngCylStrucCond RLTs, in the Structure Mass folder. From each cylinder, the masses (in grams) are:

Mass in g	**Cylinder 1**	**Cylinder 2**	**Cylinder 3**	**Cylinder 4**
Cylinder Wall Mass	758	756	756	758

Head Mass total	554	554	554	554

After subtracting these values from the appropriate parts, the ThermalMass parts will have the following attributes:

Mass in g	**Cylinder 1**	**Cylinder 2**	**Cylinder 3**	**Cylinder 4**
Block	5093.221	2189.073	2148.193	4761.983
Head	3247.771	1943.901	1940.492	2857.052

The completed model can be found in Water-Jackets-Tutorial-Calibration.gtm.

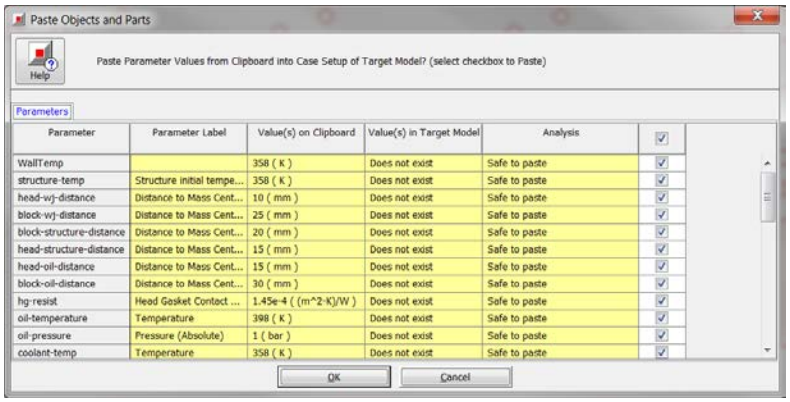
1.8 Integration with a Fast-Running Engine Model

So far, in-cylinder boundary conditions, obtained from a previously-run engine model, are imposed on the thermal model. This is necessary for the calibration process and also useful for running steady state simulation if the main focus is on heat distribution and cooling system analysis. For transient investigations like studies of the engine warm up behavior and the effects of different cooling strategies on the engine performance and fuel consumption, this approach is no longer recommended. Imposing static in-cylinder boundary conditions represents a one way coupling and misses the direct interaction between the combustion and cooling circuits. To capture these effects, a predictive engine model can be connected to the thermal model to run a transient study.

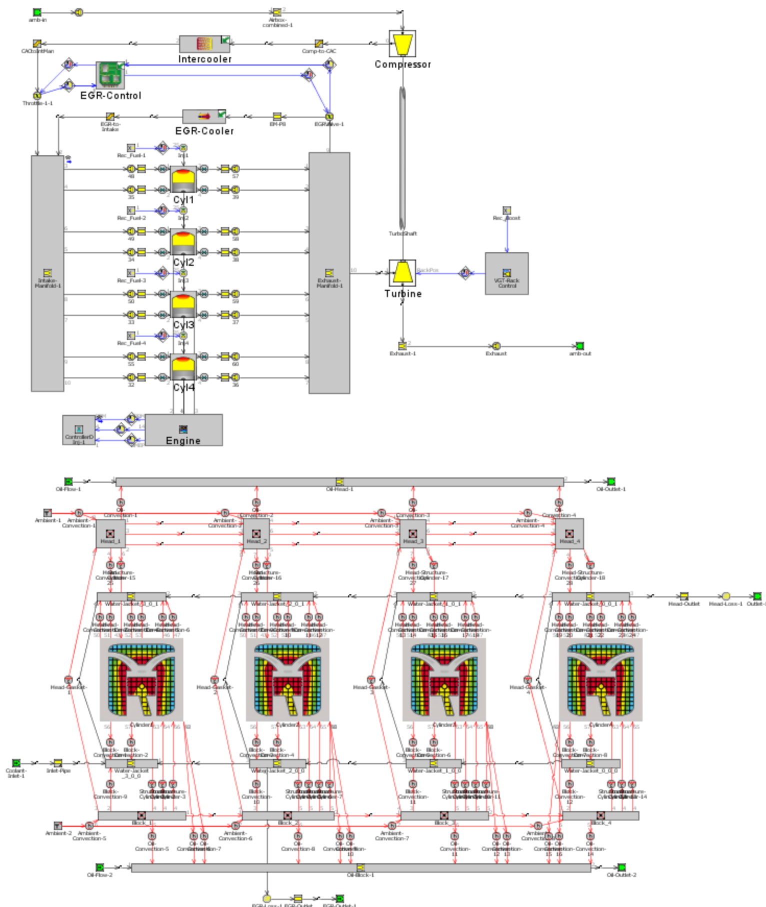
Depending on the level of detail used in the engine model, integrating the two circuits may lead to long runtimes. The cooling and thermal system portion may run in approximately real time, but the detailed engine models may run hundreds of times slower than real time. The solution is a fast-running engine model (FRM), which simplifies the engine flow circuit to enable significantly reduced run times, but still delivers the accuracy needed for thermal simulations. A tutorial for transforming a detailed engine model into an FRM can be found in Tutorial #9 of the Engine Performance tutorials group in GT-ISE under File -> Tutorials -> Modeling_Applications -> Engine_Performance.

1.8.1 Connecting the Models

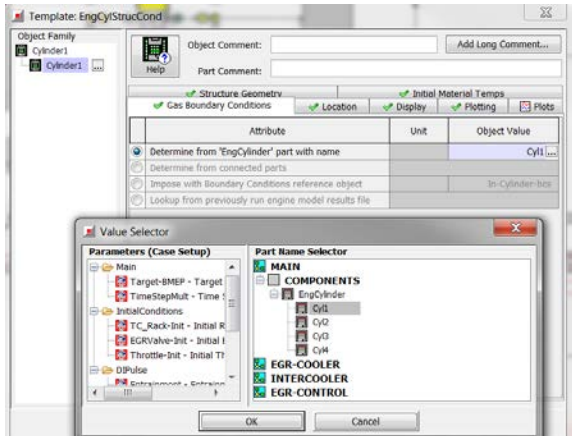
For this model, a fast-running engine model has been provided. This can be found in the Diesel_Engine_Tutorial.gtm in the tutorial directory. Open this file, and from the File menu, select the Change License option. This will allow the GT-POWER model to be switched to a GT-SUITE license which is required to run the thermal model. After the license type has been changed, copy the calibrated thermal model parts (from your tutorial file, or from Water-Jackets-Tutorial-Calibration-Final.gtm) onto the map. In the copied parts, parameters are defined which are non-existent in the target model. When the parts are copied into the new model, the Paste Objects and Parts dialog opens. This dialog is used to compare any parameters in the copied parts with existing parameters in the model. In this case, they are all safe to paste, so click OK to complete the operation.



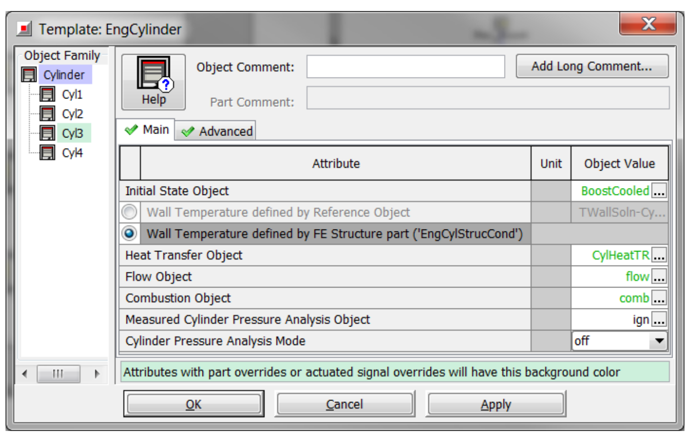
After this operation, the target model should look as follows:



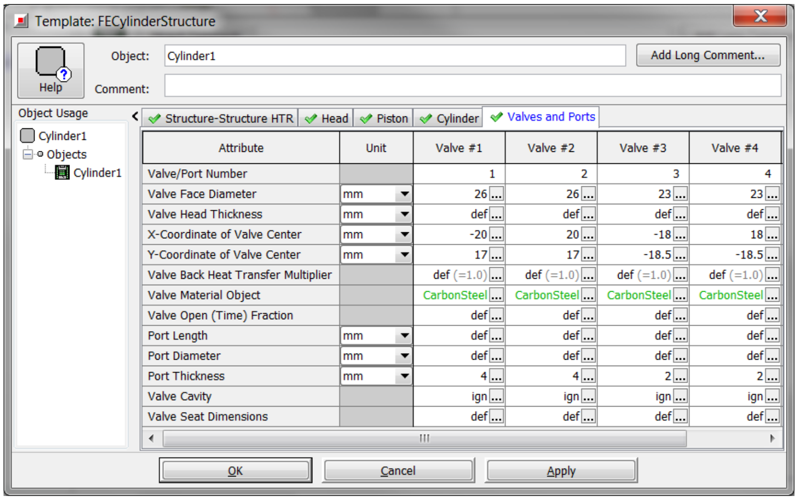
Once the parts have been imported, the first step is to establish the connection between the gas side and the structural part of each cylinder. Previously, the in-cylinder boundary conditions were manually imposed using the ‘Impose with Boundary Conditions reference object’ functionality in the Gas Boundary Conditions Tab inside the EngCylStrucCond parts. In order to obtain the boundary conditions directly from the combustion side of the Cyl1, select the "Determine from ‘EngCylinder’ part with name" option and use the value selector to point to the Cyl1.



This step must be done for the other 3 cylinders as well. The next step in integrating the models is to indicate to the combustion simulation that the wall temperatures will be obtained from the FE Structure parts. Open the Cylinder object, and select the 'Wall Temperature defined by FE Structure part ('EngCylStrucCond')' option. The Cylinder object should appear as shown:

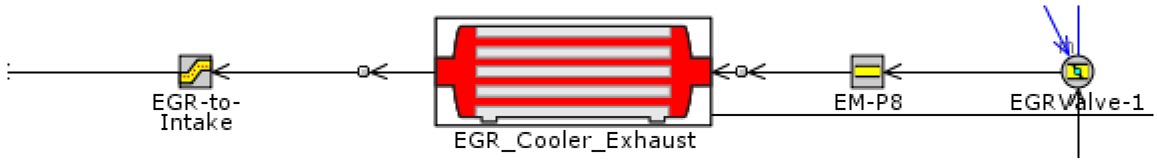


In the thermal model, the valve opening times and port geometry were added to each FECylinderStructure reference object, but with the coupled approach this information will be obtained from the engine model, therefore these attributes must be set to def in all of the EngCylStrucCond parts, as shown below:

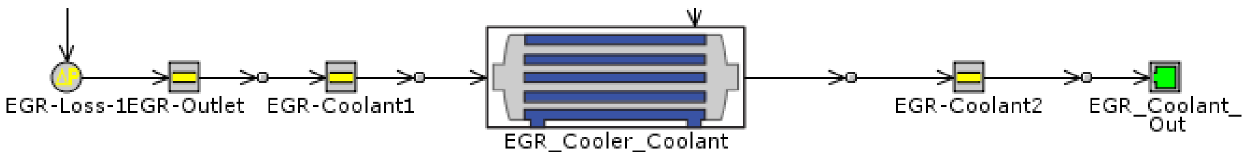


Attribute	Unit	Valve #1	Valve #2	Valve #3	Valve #4
Valve/Port Number		1	2	3	4
Valve Face Diameter	mm	26	26	23	23
Valve Head Thickness	mm	def	def	def	def
X-Coordinate of Valve Center	mm	-20	20	-18	18
Y-Coordinate of Valve Center	mm	17	17	-18.5	-18.5
Valve Back Heat Transfer Multiplier		def (=1.0)	def (=1.0)	def (=1.0)	def (=1.0)
Valve Material Object		CarbonSteel	CarbonSteel	CarbonSteel	CarbonSteel
Valve Open (Time) Fraction		def	def	def	def
Port Length	mm	def	def	def	def
Port Diameter	mm	def	def	def	def
Port Thickness	mm	4	4	2	2
Valve Cavity		ign	ign	ign	ign
Valve Seat Dimensions		def	def	def	def

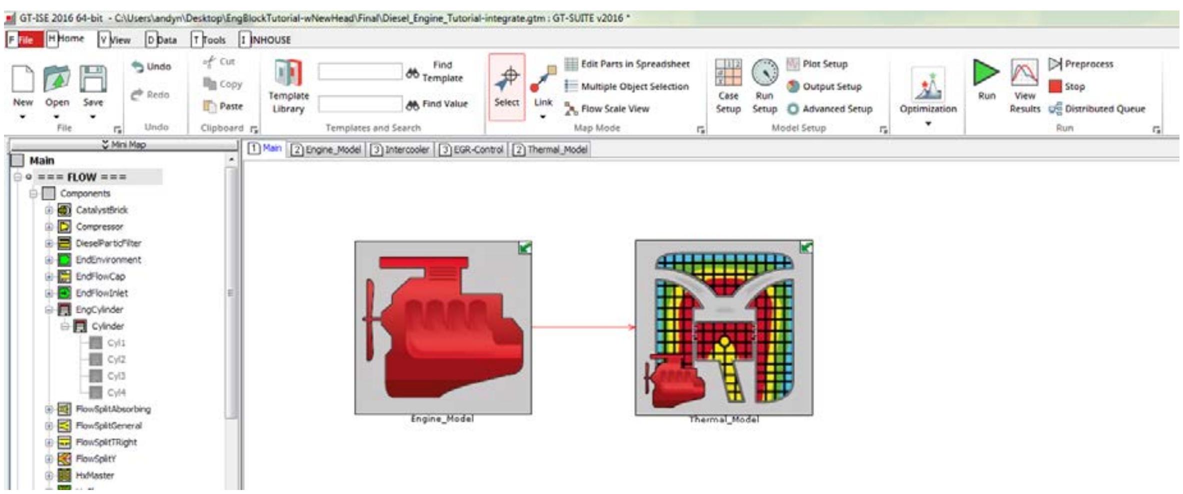
In the engine model there is an EGR-Cooler subassembly, connected to the engine circuit. The coolant side runs with imposed boundary conditions. The next step in integrating the models is now to connect the coolant side of the EGR cooler to the EGR-Outlet pipe from the thermal model. To make this connection easier, the EGR-Cooler subassembly should be combined with the main model. This can be done using the Absorb Subassembly command when right-clicking on the EGR-Cooler subassembly. Position the parts near the EGR outlet from the thermal model, and then move the gas side of the EGR cooler back up to its position in the engine model. Right-click on the EGR_Cooler_Exhaust part to rotate it, so it will fit back in the engine model part placement. See the image below for the correct placement of the EGR_Cooler_Exhaust part:



On the coolant side, remove the EGR-Outlet-1 and EGR_Coolant_In EndFlowInlet parts, along with the orifice connection between the EGR_Coolant_In and the EGR-Coolant1 part. Then, connect the orifice downstream of the EGR-Outlet part and the EGR-Coolant1 part to complete the coolant circuit. Also, in the "EGR_Experimental" HeatExchangerSpecs object, the Slave (External) Pressure Drop data object may be set to ign, since the coolant-side pressure drop is already accounted for. The model should appear as shown:



The next step is to create an internal subassembly for each subsystem, to have a bette overview of the model and to make the map look cleaner. Hold down the left mousekey and move the mouse to perform a box selection for the all of the parts belonging to the engine subsystem. Rightclick on one of the marked parts and choose ‘Create Subassembly’ in the context menu. Right click on the newly created subassembly and select ‘Part Display Settings / Choose Icon’. Click on the Choose GTI Image button and select the Engine Icon. Do the same steps to create an additional internal subassembly for the thermal model and choose the icon Engine_Structure. The map should look like this:



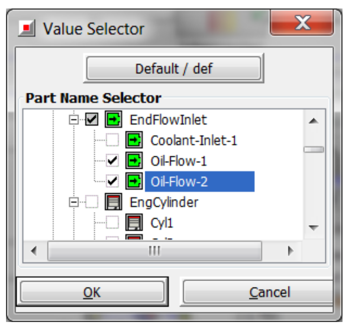
The subassemblies can be accessed by doubleclicking on the respective Icon on the map, or by using the subassembly tabs above the map.

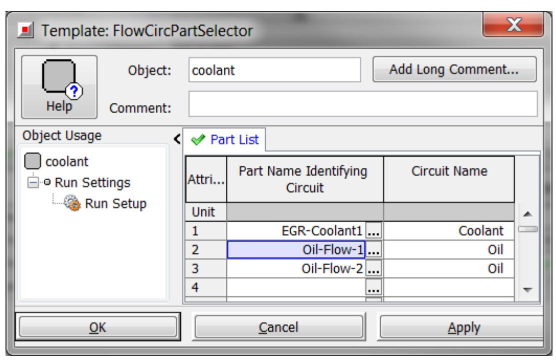
1.8.2 Run Setup for Multiple Circuits

In the thermal model, a single flow circuit was present, so the model setup was rather simple. In integrated models, the engine model must be run with the explicit solver, while the coolant circuit should be run with the implicit solver for faster run times. In the integrated model, open up Run Setup.

In the TimeControl folder, the TimeControl Flag is set to periodic due to the EngineCrankTrain present in the model. The simulation duration is set in cycles, which is suitable for steady state simulations. When following a transient profile, it is typically more convenient to specify the simulation duration in the time domain.

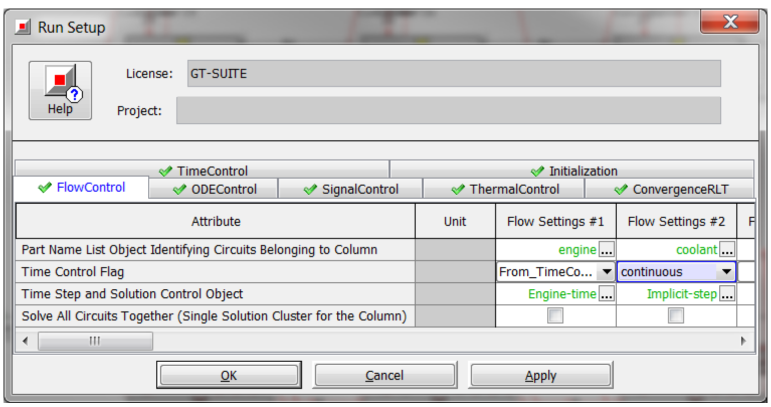
In the FlowControl folder, two separate settings have been defined. The first column is used for the engine model, with the explicit solver selected. The second column was already set up for the EGR coolant circuit in the original engine model. Since the integrated coolant model can be run with similar settings, this will be suitable for the current model. The first attribute contains a reference object which lists the parts that these settings should be applied. Double click to open the coolant reference object, change the listed Circuit Name from Coolant-EGR to Coolant and also add the Oil circuit to the part list. Use the value selector and select the two EndFlowInlets parts Oil-Flow-1 and Oil-Flow-2 which represent the oil circuit and enter Oil as the circuit name.





Further differences when running an integrated model are illustrated in the Time Step and Solution Control Object. Open the reference object labeled "Implicit-step." When an implicit circuit is run along with a periodic engine circuit, the time step for the implicit circuit is definied in terms of the crank angle rotation instead of in seconds. For this model, a time step of 720 degrees is used, which means that the coolant circuit will take one time step per cycle of the detailed engine circuit. It is the maximum time step size allowed when running the cooling circuit periodically. At higher engine speeds, the time step will be a bit smaller than the standard 0.1 seconds used when running a "continuous" circuit.

For steady-state simulations, the periodic time step measurement can be used for both circuits. As soon as the engine speed starts to vary during simulation, for example when running a transient driving cycle, different settings are recommended. The Improved Solution Sequence for Multi-Circuit Models should be turned on by checking the box in the TimeControl Tab in RunSetup. This enablesthe circuits to be defined independently. The engine circuit can be run periodically, and the coolant circuit can be run using continuous time step measurement. If this option is selected, the Time Control Flag in the TimeControl folder can remain periodic, but the FlowControl folder should be changed so that the Time Control Flag for column #2 is set to continuous as shown below:

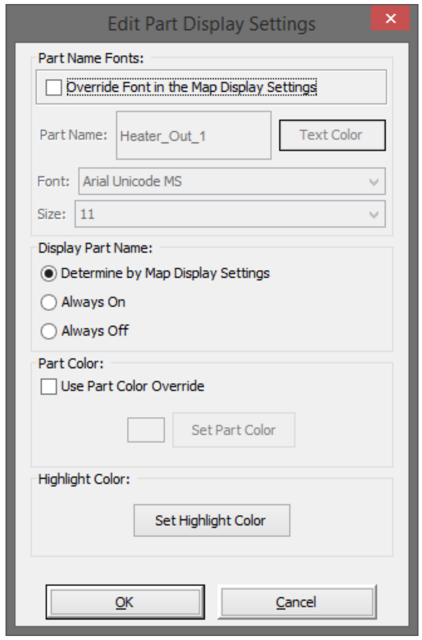


In addition, the Time Step size in the Implicit-step reference object will now be defined in seconds. This needs be changed to "def", rather than the value of 720 (seconds).

The complete model is stored in the tutorial directory with the name Thermal-Model-Integrated-Final.gtm.

1.8.3 Coloring a Circuit (Optional)

Coloring a circuit is recommended to easily distinguish one fluid circuit from another in a model. This can be done by right-clicking on any flow volume part in the model, picking the Select Circuit for Part option, and then right-clicking on a part again to select Part Display Settings. This will launch the Edit Part Display Settings dialog.



Enable the checkbox for the Use Part Color Override, and select the desired color for the flow circuit. Once a color is selected, click OK on the Edit Part Display Settings dialog to assign the color to the circuit (flow volumes) selected.

1.9 Integration with Other Circuits

To build a complete model for studying the warm-up of a vehicle, other systems must be included in the model. These include the coolant system, the oil system, and some representation of the vehicle underhood for heat rejection. A vehicle model can also be coupled to the model to provide a more accurate prediction of the engine speed and torque requirements to complete the desired driving cycle. One example of a complete integrated model is the Transient_Warmup_Drive_Cycle model, in the Cooling_Engine example group. This model contains a fast running engine model, the engine block, cooling system, oil system, vehicle drivetrain, controls and friction calculation. The detailed methodology for building each of these other systems can be found in other GT-SUITE tutorials, but a discussion of the systems as they relate to the current model follows.

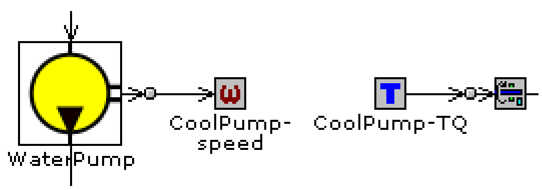
1.9.1 Integrating the Coolant System

The coolant system connections to the current model are straightforward. In a vehicle-level simulation, a simple representation for the engine is used to represent the combined block and head water jackets.

That should be replaced with the current engine model. The outlet from the pump can be connected directly to the inlet of the engine block, and the outlets from the block system should be connected to the respective pipes and flowsplits in the coolant model.

The water pump torque can be accounted for in a few different ways, depending on the pump type. For a mechanical pump, the engine speed should be imposed on the pump through a mechanical connection accounting for any gear ratio. This can be accomplished by first creating a shaft connection to the EngineCrankTrain at port 81 (Auxiliary Torque), using a GearConn to provide the gear ratio. The gear ratio is calculated using the formula: $GearRaio = \frac{\omega_{input}}{\omega_{output}}$, where the input is the incoming link, and the output is the outgoing link from the connection. On the end of this shaft should be a Torque component to provide the torque feedback to the Engine. The Torque value is obtained from an RLTDepdence using the Average Torque RLT from the pump. The water pump is driven using a SpeedBoundaryRot where the Imposed Speed attribute is obtained from an RLTDependence using the Average Speed RLT from the shaft on the pump side of the GearConn.

This indirect connection is typically used instead of a direct connection to the engine to increase the stability of the flow solution. Since the engine speed is fluctuating within one cycle, even at a "steady" cycle averaged speed, the changes in engine speed can cause some instability through the pump. Using the average speed engine speed for the pump provides a more stable solution, and the torque imposed back on the engine will be appropriate for the pump operation at a larger time step.



For an electric water pump, the Torque component should be dependent on the Average Torque RLT from the pump, but it can be connected directly to the EngineCrankTrain instead of through the shaft and gear connection. The SpeedBoundaryRot should be used to drive the pump, but instead of specifying the speed through an RLTDependence, the speed can be imposed directly in the SpeedBoundaryRot or by using a controller.

1.9.2 Integrating the Oil System

The oil system connections to the current model are similar to the coolant circuit. The "oil film" volumes that were created to represent the oil in the head and engine block should be connected to the rest of the oil circuit downstream of the head and block bearings, respectively. The typical assumption for thermal warm-up models is that the heat transfer from the block occurs in the oil film volumes, and not in the passages leading to the bearings. This assumption simplifies the connections to the thermal model but may require some adjustment of the heat transfer coefficient in the oil film volume to calibrate the model. The friction from the bearings, if this is available on the component level, should be added as a source term to the oil film volumes.

For thermal warm-up models, a detailed oil circuit with predictive JournalBearingFlow parts is not recommended. The predictive model requires a small time step, which will significantly increase the run time for a driving cycle model. Instead, the JournalBearingFlow template offers a "Mean Value" mode; which looks up the bearing behavior in a map to speed up the computation time. This mode should be used if a detailed lubrication circuit is desired. After the oil passages and bearings, the oil would flow into the "oil film" volumes created in the thermal model, and then drain back to the oil pan volume. The oil piston jets could be included as separate flow volumes, connected to Link ID 42 (the Piston Oil surface) on the EngCylStrucCond instead of connecting the general oil film volume to this boundary.

Another approach is to simplify the oil circuit so the detailed flow through every passage and bearing is not calculated, but the overall flow pattern and oil volume is conserved. The oil passages in the block and head would be combined into larger volumes, and FlowMap components to represent the lumped bearing losses. After the combined oil passages and bearings, the oil would flow into the "oil film" volumes created in the thermal model, and then drain back to the oil pan volume.

The oil pump should be connected in a similar way to the water pump. If a PumpFlow is used instead of the standard pump, the total power can be converted to a torque using the engine speed and imposed in the Torque component connected to the EngineCrankTrain.

The heat rejection from the oil circuit can be handled in several ways, depending on the type of cooler used. For an oil-to-water cooler, the HxMaster part can be connected to the oil circuit and the HxSlave part can be connected to the coolant circuit in the appropriate branch. This is similar to the EGR cooler in Section 1.8.1. For an oil-to-air cooler, the air side can be modeled as discussed in the next section.

1.9.3 Integrating the Underhood System

For each of the heat exchangers in the model, external boundary conditions are required. In a basic model, the air flow over each heat exchanger could be imposed independently. To model the interaction between heat exchangers, and the effects of the heat exchanger and fan placement, COOL3D can be used to construct a detailed model of the underhood system. The HxMaster and HxSlave parts in the cooling and engine systems can be replaced by their COOL3D counterparts. The same HeatExchangerSpecs template is used to define the heat exchanger in both applications, so the data can be directly transferred to COOL3D using a .gto file.

Note that the heat exchangers built in Cool3D will have two discretization lengths for the external side. One discretization direction is parallel to the internal flow, and one is perpendicular to the internal flow. The discretization parallel to the internal flow direction will also affect the subvolume size in the internal tubes of the heat exchanger, so for the Charge Air Cooler, this value should be carefully matched to the discretization length of the rest of the engine model. For more details on building the COOL3D model, please see the Cooling_Thermal_Management tutorials 3 and 4.

When the COOL3D model has been completed, or if the heat exchangers will be modeled independently, the internal circuits (engine, coolant, and oil) should be connected to the HxMaster or internal side of the MatrixHx part. If COOL3D is used to model the air side, the external volume of the heat exchangers will be automatically connected as the output of the COOL3D discretization. If the heat exchangers will be modeled independently, a simple circuit can be built with an EndFlowInlet to provide the air flow, flowsplits to model the upstream and downstream volumes, and an EndEnvironment to provide the downstream pressure.

1.10 Friction Modeling for Thermal Warmup

Modeling the friction losses is very important to the behavior of a thermal warm-up model. The increased friction loads on the engine under cold start conditions cause both an increase in the heat transferred to the structure, and an increase in the fuel consumption of the engine. The friction model used must capture this behavior, typically based on the oil temperature, for accurate warm-up behavior.

1.10.1 Using Friction from Strip-Down Tests

One method for capturing the relationship between the oil temperature and the friction torque on the engine components is to measure the effect directly. In a "strip-down" test, the motoring torque of the engine is measured at various stages of engine disassembly. This allows the friction contribution of different components to be measured as they are removed and the total torque is recorded. Important groups for measurement include the Valvetrain, the Piston group, the Crankshaft bearings, and any balance shafts or other oil-lubricated components. The tests at each stage of engine components can be performed at varying engine speeds and oil temperatures to generate a map of the friction torque for each component.

In the GT-SUITE thermal model, these maps can be implemented as controls components. Sensors to measure the oil temperature and engine speed in the model can be used as inputs to the maps to provide the friction values. After these values have been calculated, they should be used in two places. The first is to convert the Torque (or MEP, depending on the map units) to Power based on the engine speed and impose this power as a heat load on the thermal model. The power calculated from the valvetrain should be imposed on the oil volume in the head, the power calculated from the connecting rod bearings and main bearings should be imposed on the oil volume in the block, and the power calculated from the piston group should be imposed on the EngCylStrucCond parts at link 40 (Cylinder Ring and Skirt Friction).

The second place the values should be used is in the EngineCrankTrain part, as the FMEP value. The individual component friction values can be combined, and then converted to an MEP value (depending on the measurements contained in the maps) and used to actuate the FMEP in the EngineCrankTrain. For an example of how to connect these friction maps to a detailed thermal model, see the Cooling_Engine\Transient_Warmup\Transient_Warmup_Drive_Cycle example model.

1.10.2 Using Theoretical Models

If detailed friction measurements are not available, the friction values will have to be modeled using a different approach. One popular friction model for diesel engines is the Schwarzmeier-Reulein model. This model is available as a controls compound in GT-SUITE, and it provides an overall FMEP value based on some engine dimensions and a reference friction value from measurements. The model is valid down to 293 K, and allows for multipliers to the friction terms to adjust the model performance to match test data points if they are available.

For gasoline (SI) engines, the Fischer friction model is provided in GT-SUITE as a controls compound. This model requires no geometrical data about the engine, but it does require two measurement points at varying engine speeds, warm conditions (90 C), and 0 bar BMEP. From these two points, the FMEP dependence is calculated as a function of speed and temperature (assuming a function for the temperature dependence). There are no built-in multipliers for the Fischer model; however additional controls components could be connected to the output to match test data points.

These controls compounds provide the total FMEP, which can be connected directly to the EngineCrankTrain as discussed in the previous section. For the heat contribution to the oil and structure, the FMEP value must be converted to a power and then divided among the various groups. A good starting place for this division would be 50% of the total friction value to the piston group, 30% to the valvetrain, and 20% to the crankshaft bearings, although this can change depending on the engine configuration. Note that these models do not calculate the accessory loads, so those would need to be connected separately.

TUTORIAL 2: Definition and Use of 'EngCylStrucCustom' in GT-SUITE v2016

Note that this is not a formal tutorial, as no supporting files are available. It is simply a discussion of a new feature available in v2016. A full tutorial will be provided in a future build of GT-SUITE.

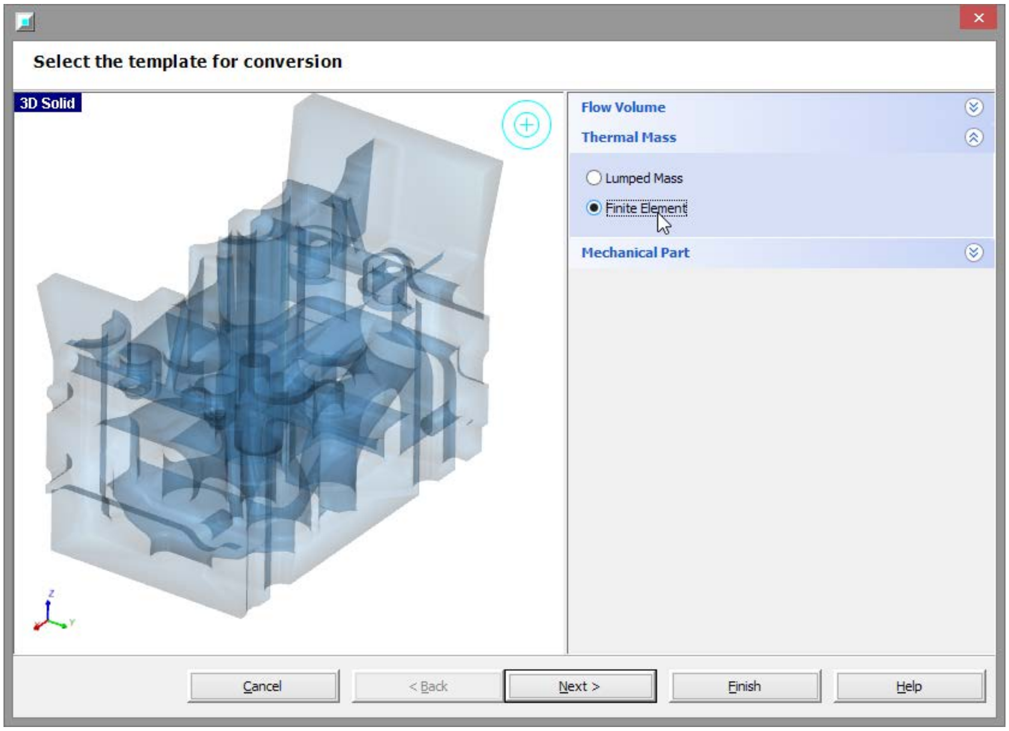
2.1 Steps for Building the EngCylStrucCustom in GT-SUITE v2016

2.1.1 Import the CAD geometry to GEM3D

The CAD geometry to be converted to a finite element mesh should be imported to GEM3D. The recommended parts to import are the engine block, head, piston, and the valves for a single cylinder. It is ideal if the parts are all positioned in the correct locations relative to another, but different coordinate systems can be accounted for by defining the local origin of each component.

2.1.2 Convert the imported geometry to a Finite Element component

Once the geometry has been imported, it has to be converted to a GEM3D component. For the EngCylStrucCond, the block and head must be divided up cylinder-by-cylinder and converted individually. Right-click on the Solid Shape and select the "Convert Shape to Component" option. Select the Finite Element option from the Thermal Mass menu.



The minimum and maximum element sizes can be given during the conversion process. Default values have been given that work well for most geometry. Depending on the complexity of the geometry, the meshing process may encounter some errors. If the full mesh does not appear after the conversion element size has been given and the conversion process has been completed, it may be necessary to use the "De-convert Component" tool from the Slicer menu and re-try the conversion with a smaller element size.

2.1.3 Create the surface ports on the GEMThermalFE component

Certain boundary surfaces are required for the EngCylStrucCustom. These surfaces are listed below for each template:

Cylinder Block

- Either **Cylinder Bore (Gas and Oil)** or **Liner Contact** (depending on if the liner is a separate FE mesh or not)
- **Block Head Gasket Contact**
- Optional: Block Coolant 1->10, Block Oil 1->10, Block Ambient 1->5, Block Custom 1->10

Cylinder Head

- Either **Head Combustion Chamber Gas Side (all zones)** or **Head Combustion Chamber Gas Side 1, 2, 3**

- **Head Gasket Contact Area**
- Ports are not required in the Head, but if the **Port 1->5 Gas Side** surfaces are used, then the **Valve Seat Contact Area 1->5** should also be defined. The **Valve Guide Contact Area 1->5** ports, if they are used, also require both of these additional surfaces.
- Optional: Upstream Intake Port Gas 1->5, Downstream Exhaust Port Gas 1->10, Head Coolant 1->10, Head Ambient 1->5, Head Oil 1->10, Head Custom 1->10

Cylinder Liner

- **Cylinder Bore (Gas and Oil)**
- **Liner Block Contact**
- Optional: Liner Head Gasket Contact

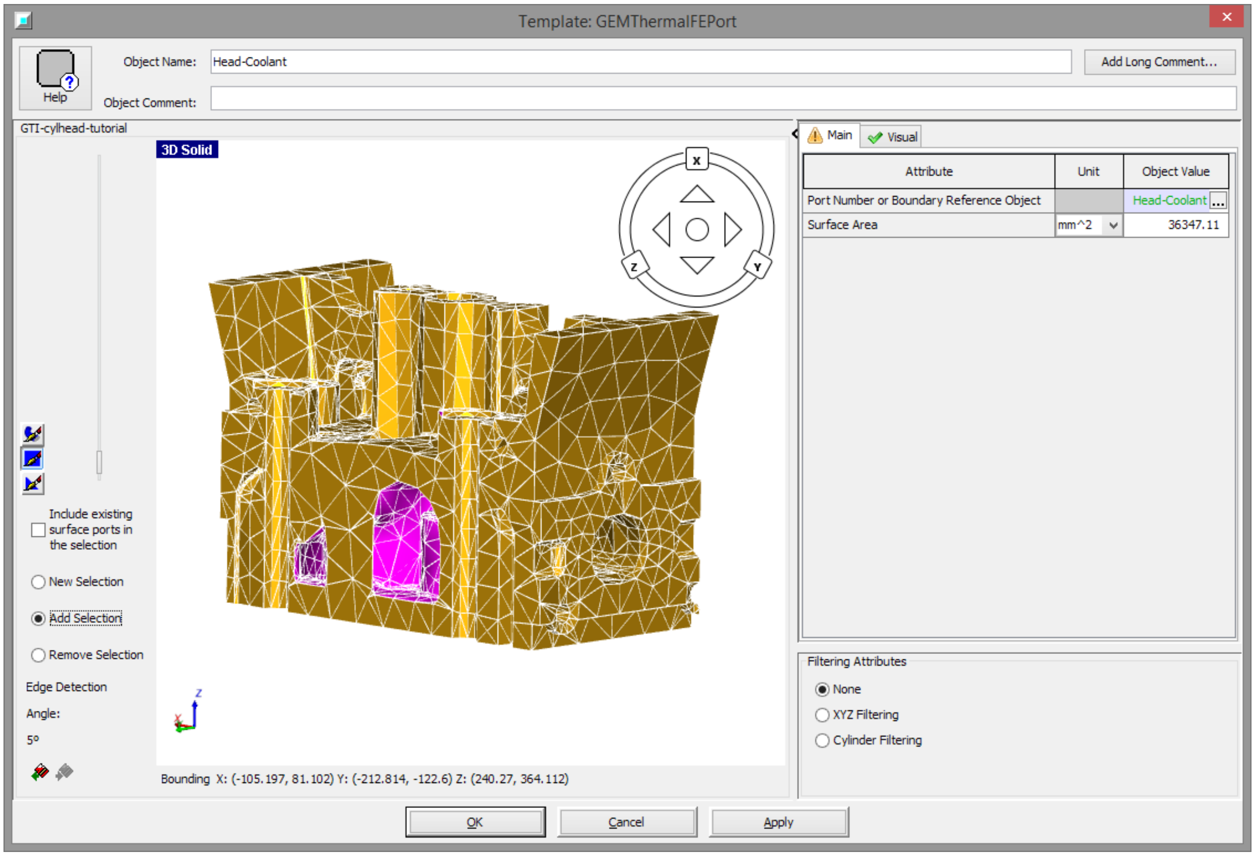
Piston

- Either **Comb Gas 1-3 (Auto)** or all three **Comb Gas 1**, **Comb Gas 2**, and **Comb Gas 3**
- **Oil 1**
- **Cylinder Contact**
- **Ring Groove 1**
- Optional: Oil 2-3, Ring Groove 2-5

Valve

- **Valve Front (Combustion Chamber)**
- **Valve back**
- **Valve Seat Contact Area**
- **Valve Guide Contact Area**

To create a surface on the Finite Element component, right-click on the GEMThermalFE part and select the Thermal FE Port option. The following window should appear:



On the left hand side of the window, the slider bar controls the angle tolerance for the selection of the surface triangles. Clicking on a surface in the 3D window (middle section) will either: create a new selection, add to the current selection, or remove from the current selection according to the radio button selected. On the right hand side, the Filtering Attributes (at the bottom) can be used to limit the surfaces that will be selected with a new mouse-click in the 3D window. The **Port Number or Boundary Reference Object** attribute can be used to assign a number or name that will be used in the Port ID for the FEMesh3D. Careful naming of the Port IDs will make the assigning of surfaces in the EngCylStrucCustom much easier. If a name is entered, the Boundary Reference Object does not need to be defined, the name will be used in the mesh and connections can be specified in GT-ISE.

2.1.4 Create the .gtm file

Once the surfaces have been created on the GEMThermalFE parts, the model is ready to be exported to GT-ISE. The File > Export GTM command will create the GT-ISE parts from the GEM3D components. The GEMThermalFE component will be discretized to a ThermalFiniteElement part in the GT-ISE model which calls the FEMesh3D. The GEMThermalFEPort features that were created will be added to the Ports folder in the FEMesh3D.

2.1.5 Create a new EngCylStrucCustom object

In GT-ISE, open the exported .gtm file and then import the EngCylStrucCustom template from the Template Library. This template will reference the FEMesh3D objects that were created in GEM3D, map the Port IDs to defined surfaces in the cylinder structure, and position the parts so that conduction connections and plots can be created.

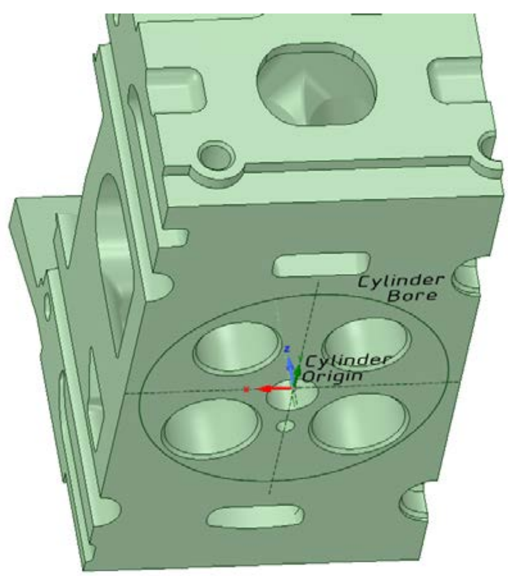
Before creating the reference object to define the cylinder structure, some basic information can be provided in the template. The Cylinder Geometry Object can be referenced from an existing engine model, or a new one can be created with the basic engine dimensions. The initial material temperatures can also be defined. The Gas Boundary Conditions can be provided, for a simple test case the "Impose with Boundary Conditions" option is recommended. Using this option will allow for a test model to be run to confirm the setup, before the structure is used in an engine or cooling system model.

2.1.6 Create an FECylStrucCustom object

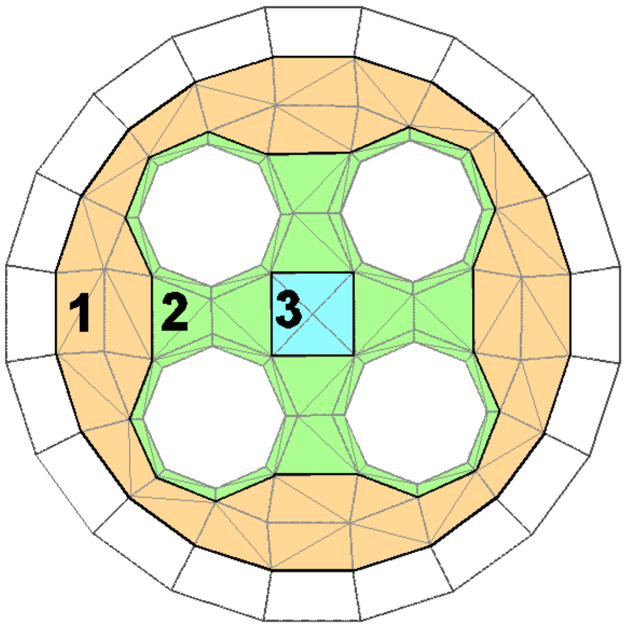
The FECylStrucCustom is the template used for organizing the meshes that form the cylinder structure. It references the EngCylBlockFE, EngCylHeadFE, EngCylPistonFE, and EngCylValveFE (with the option EngCylLinerFE if the cylinder liner is a separate material). Each of these reference objects point to FEMesh3D objects to define the mesh, defines the reference point for the local coordinate system, and also provide a mapping for the surfaces in the mesh to the named surfaces in the EngCylStrucCustom.

Head - EngCylHeadFE

The local origin is defined as the center of the cylinder bore, on the top surface of the block (typically the bottom surface of the head. The surface descriptions are below the origin diagram:



- **Head Combustion Chamber Gas Side (all zones):** This surface represents the area on the head mesh that is exposed to the combustion gas. It will be automatically divided into zones 1, 2, and 3 according to the diagram below. The actual zone sizes and shapes may be different depending on the mesh resolution used in the 'FEMesh3D' and the shape of the surface selected on the mesh.

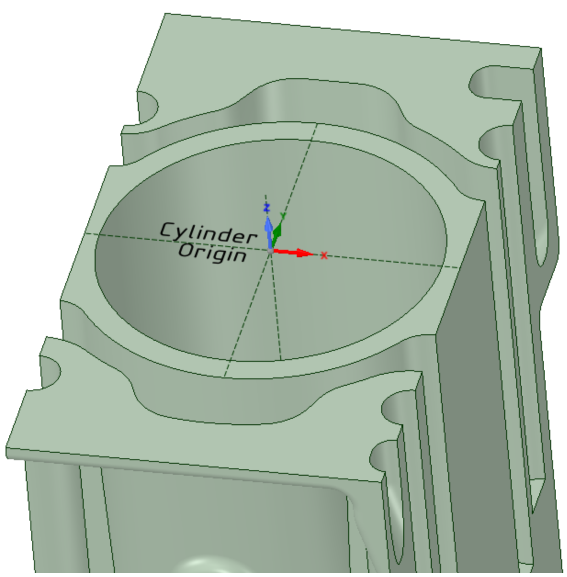


- **Head Combustion Chamber Gas Side 1, 2, 3:** If the automatic division of zones 1, 2, and 3 on the combustion side of the head fails or does not match the desired zone 1, 2, and 3 areas, then these ports can be used to manually select the three zones.
- **Head Gasket Contact Area:** The bottom surface of the head that is in contact with the block.
- **Port 1->5 Gas Side**: The internal surfaces of the head that are in contact with the intake and exhaust ports. These surfaces should correspond to the port pipes and flow splits that are directly connected to the cylinder in the engine model.
- **Upstream Intake Port Gas 1->5**: Surfaces that are available for connecting gas flow volumes upstream of the cylinder, inside the head.
- **Downstream Exhaust Port Gas 1->10**: Surfaces that are available for connecting gas flow volumes downstream of the cylinder, inside the head
- **Valve Seat Contact Area 1->5**: The surfaces on the head that will be in contact with the valves when they are closed. A conduction connection will automatically be created between the valve and the valve seat surface, according to the valve opening time. If these surfaces are selected, the corresponding **Port Gas Side** surfaces should also be defined.
- **Valve Guide Contact Area 1->5:** The surfaces on the head that are in contact with the valve stem. A conduction connection will be created between these surfaces and the valve stem.

- **Head Coolant 1->10:** Surfaces that are available for connection to coolant boundary conditions. The heat rates from these surfaces will be summed to create the "Heat Transfer Coolant to Head" RLT. Note that each surface can only connect to a single boundary condition, so if the water jacket has been divided into multiple flow volumes a surface should be created for each.
- **Head Ambient 1->5**: Surfaces that are available for connection to ambient boundary conditions. The heat rates from these surfaces will be summed to create the "Heat Transfer Ambient to Head" RLT. Note that each surface can only connect to a single boundary condition.
- **Head Oil 1->10**: Surfaces that are available for connection to oil boundary conditions. The heat rates from these surfaces will be summed to create the "Heat Transfer Oil to Head" RLT. Note that each surface can only connect to a single boundary condition, so if the oil passages and film have been divided into multiple flow volumes a surface should be created for each.
- **Head Custom Port 1->10**: Surfaces that are available for connection to custom boundary conditions. These can be convection or conduction boundaries. The heat rates from these surfaces will be summed to create the "Heat Transfer Other to Head" RLT. Note that each surface can only connect to a single boundary condition.

Block - EngCylBlockFE

The local origin is defined as the center of the cylinder bore, on the top surface of the block. Surface descriptions are below the origin diagram:

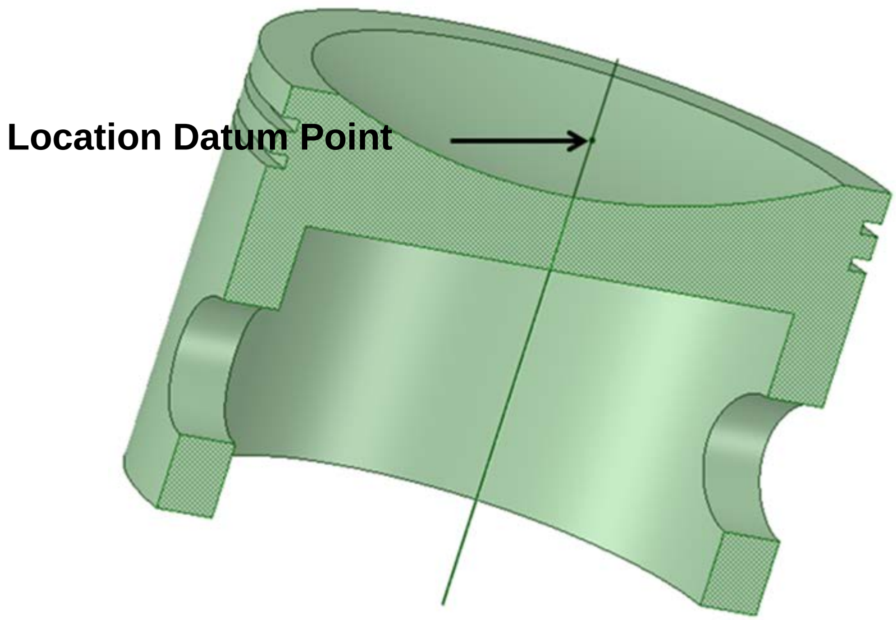


- **Cylinder Bore (Gas and Oil):** This surface represents the inside of the cylinder bore, if it is exposed directly to the combustion gas and oil beneath the piston. If this surface is used, the Liner Contact surface should not be used.

- **Liner Contact:** The surface that is in contact with the liner mesh. A conduction connection will be created between this surface and the corresponding surface on the outside of the liner.
- **Block Head Gasket:** The top surface of the block that is in contact with the head.
- **Block Coolant 1->10:** Surfaces that are available for connection to coolant boundary conditions. The heat rates from these surfaces will be summed to create the "Heat Transfer Coolant to Cylinder Block" RLT. Note that each surface can only connect to a single boundary condition, so if the water jacket has been divided into multiple flow volumes a surface should be created for each.
- **Block Oil 1->10**: Surfaces that are available for connection to oil boundary conditions. The heat rates from these surfaces will be summed to create the "Heat Transfer Oil to Cylinder Block" RLT. Note that each surface can only connect to a single boundary condition, so if the oil passages and film have been divided into multiple flow volumes a surface should be created for each.
- **Block Ambient 1->5**: Surfaces that are available for connection to ambient boundary conditions. The heat rates from these surfaces will be summed to create the "Heat Transfer Ambient to Cylinder Block" RLT. Note that each surface can only connect to a single boundary condition.
- **Block Custom Port 1->10**: Surfaces that are available for connection to custom boundary conditions. These can be convection or conduction boundaries. The heat rates from these surfaces will be summed to create the "Heat Transfer Other to Cylinder Block" RLT. Note that each surface can only connect to a single boundary condition.

Piston - EngCylPistonFE

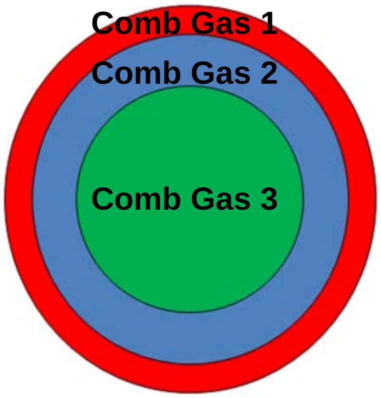
The local origin for the piston is *not* the cylinder origin. Instead, it is at the center of the bore on the top surface of the piston. The surface descriptions are below the diagram:



- **Comb Gas 1-3 (Auto)** – Use either this option OR all three of the following "**Comb Gas**" options to define the top surface of the piston that transfers heat with the combustion gas. This "auto" option is recommended for most models. However, if the piston has a significant bowl in the top surface AND the model will make use of either the "SITurb" combustion model or the "flow" heat transfer model, then it is recommended to use the three separate "Comb Gas" selections.

If the "auto" option is selected, all of the surfaces that make up the top of the piston can be combined under this single description. The solver will then automatically assign surfaces to combustion gas zones 1-3 based on the average node radius from the piston center targeting the following area ratios (they may not be exactly achieved depending on number and shape of surfaces) :

  - Comb Gas 1 = outer-most 15% of area
  - Comb Gas 2 = middle 35% of area
  - Comb Gas 3 = center-most 50% of area



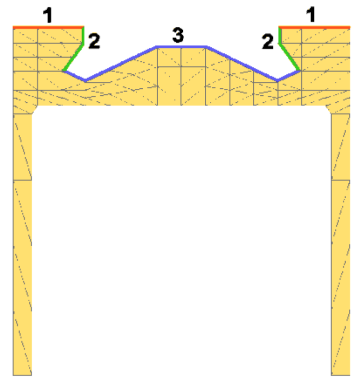
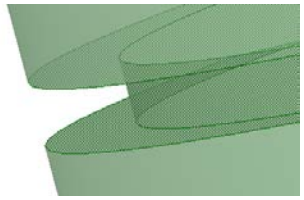


Diagram 1: Combustion Gas Auto Surfaces

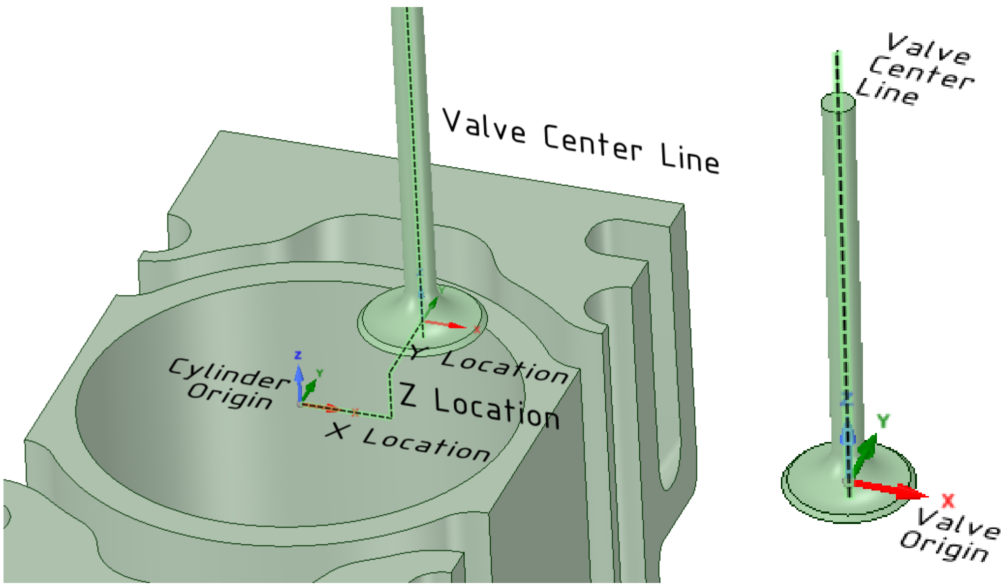
Diagram 2: Combustion Gas Surfaces with Bowl

- **Comb Gas 1 (outer)** – This alternative to the above "auto" option allows surfaces on the top of the piston to be explicitly assigned to combustion gas zone 1. In this case, the zone 1 surfaces should include the outer-most area of the piston outside of the bowl (see diagram 2 above).
- **Comb Gas 2** – This alternative to the above "auto" option allows surfaces on the top of the piston to be explicitly assigned to combustion gas zone 2. For pistons with a bowl shape, zone two should be the "outer walls" of the bowl as shown in the diagram 2 above.
- **Comb Gas 3 (center)** – This alternative to the above "auto" option allows surfaces on the top of the piston to be explicitly assigned to combustion gas zone 3. For pistons with a bowl shape, zone 3 should be the "floor" of the bowl as shown in the diagram 2 above.
- **Oil #** – Defines a contact surface for convection heat exchange with the engine lubrication oil. One oil surface (**Oil 1)** is required, and up to three total oil surfaces may be defined. This allows for the boundary conditions for piston oil jets or internal oil galleries to be separate from the overall "oil splash" surface if desired.
- **Cylinder Contact** – Defines the surface as a contact surface for conduction heat exchange with the cylinder wall. Any friction heat will also be applied as a source term to this surface. At least one surface must be assigned to this option.
- **Ring Groove #** - Defines the surfaces that make up a ring groove. The surfaces assigned to each ring groove should include the top and bottom faces of the groove as well as the inside face, as shown below. The solver will automatically match up surfaces within the groove to the surfaces on the FE ring (the ring is automatically created by the solver to "fill" the groove).
-



Valves - EngCylValveFE

The valve positioning is comprised of two coordinate systems. The local valve coordinates for each valve are defined in the EngCylValveFE, with the origin at the valve face and the Z axis aligned along the valve stem. In the FECylStrucCustom, the position and orientation of each valve within the cylinder is defined relative to the cylinder origin. This enables the same values to be used for cylinders where the valves are in the same relative positions, such as one bank of a V engine or an inline engine.



- **Valve Front (Combustion Chamber):** The portion of the valve that will be exposed to the in-cylinder boundary conditions
- **Valve Back:** The portion of the valve that will be exposed to the port gas when the valve is closed.
- **Valve Seat Contact Area:** The surface on the valve that is in contact with the head when the valve is closed. A conduction connection will be created between this surface and the corresponding **Valve Seat Contact Area** surface on the head.
- **Valve Guide Contact Area:** The surface on the valve that is in contact with the valve guide as the valve moves. A conduction connection will be created between this surface and the corresponding **Valve Guide Contact Area** surface on the head

Liner - EngCylLinerFE

The use of a liner mesh is optional, and should be used in cases where the liner is a different material than the rest of the block. The local origin for the liner is the same as the block (center of the cylinder bore, on the top surface of the block). The surfaces available are as follows:

- **Cylinder Bore (Gas and Oil):** This surface represents the inside of the cylinder bore, if it is exposed directly to the combustion gas and oil beneath the piston.
- **Liner-Block Contact:** The surface that is in contact with the mesh defined in the 'EngCylBlockFE' (referenced in the 'FECylStrucCustom'). A conduction connection will be created between this surface and the corresponding surface on the inside of the block.
- **Liner Head Gasket Contact:** The top surface of the block that is in contact with the head.

2.1.7 Test the Structure

Once the mesh parts and the boundary surfaces have been defined, some simple tests should be run to confirm the finite element data is correctly defined. Due to the significant increase in run-time, it's not recommended to run transient thermal models until the steady-state model is verified. Simple boundary conditions (gas temperatures and heat transfer coefficients on the in-cylinder side) should be imposed, and some static coolant and oil boundary conditions should be defined. The model should be run for a brief duration with the steady thermal solver to identify any errors with the mesh positions or connections. The Component FE Heat Balance, Component FE Solution Temperatures, and Component Heat Transfer Zones plots are helpful to verify the correct setup. After this test is successful, some steady-state heat transfer models are recommended to confirm that the heat distribution is correct. Then, a transient model can be run.