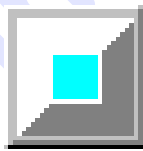


# GT-SUITE

## ***GEM3D User's Manual and Reference Manual***

VERSION 2017



by

***Gamma Technologies***

Copyright 2016 © Gamma Technologies LLC. All rights reserved.  
All information contained in this manual is confidential and cannot be reproduced or  
transmitted in any form or by any means, electronic or mechanical, for any purpose, without  
the express written permission of Gamma Technologies LLC.

## GT SUPPORT

- TELEPHONE: (630) 325-5848
- FAX: (630) 325-5849
- E-MAIL: [support@gtisoft.com](mailto:support@gtisoft.com)
- Web Address: [www.gtisoft.com](http://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**

<b>CHAPTER 1: Introduction .....</b>	<b>1</b>
<b>CHAPTER 2: GEM3D Menu Items.....</b>	<b>2</b>
File Menu.....	2
Home Menu .....	3
View Menu .....	6
Flow Menu.....	9
Thermal Menu .....	13
Mechanical Menu .....	15
Convert Menu .....	16
Dimension Menu .....	20
Tools Menu.....	22
Component Menu .....	23
<b>CHAPTER 3: GEM3D Dialogs.....</b>	<b>27</b>
Assembly Rotation - Rotates an entire assembly of connected components .....	28
Boundary Manager - Manages the connections at the boundaries (inlets and outlets) of the model.....	29
Case Setup - Organizes cases and defines parameters.....	30
Center of Gravity - Calculates the center of gravity of components .....	32
Component Rotation - Rotates a component or assembly relative to the connected component .....	33
Convert Shape to Component.....	34
Convert Shape Wizard: Pipe Conversion.....	36
Convert Shape Wizard: Perforate Feature Conversion .....	38
Convert Shape Wizard: Flowsplit Conversion 3 Ports.....	40
Convert Shape Wizard: Flowsplit Conversion 4 Ports.....	42
Convert Shape Wizard: Flowsplit Conversion .....	43
Convert Shape Wizard: Miter Bend Conversion.....	44
Convert Shape Wizard: Shell Conversion.....	45
Convert Shape Wizard: System of Pipes and Flow Splits.....	50
Convert Shape Wizard: System of Flow Splits .....	53
Convert Shape Wizard: Pipe-T-Pipe Conversion.....	54
Convert Shape Wizard: Multiple Pipes (Parallel) .....	57
Convert Shape Wizard: 3D Tank .....	59
Convert Shape Wizard: Thermal Mass Conversion .....	60
Convert Shape Wizard: Thermal Finite Element Conversion.....	61
Convert Shape Wizard: Mechanical Finite Element Conversion.....	62
Convert Shape Wizard: Mechanical Rigid Body Conversion.....	63
Cross Section Editor - 2D editor used to create and modify custom cross sections .....	65
Cutting Plane Control Window - Controls the direction and orientation of the cutting plane.....	68
Export Image - Export graphical view(s) to an image file.....	69
Export Model - Export model file for use in GT-ISE.....	70
Export STL - Export model to an STL file.....	83
Export ACIS - Export model to an ACIS file .....	84
Filter Shapes - Filter shapes.....	85
Import 3D - Import 3D file .....	86

Import STL - Import STL file.....	88
Import STL as Surface - Import an entire STL file from its surfaces.....	89
Import STL as Cross Sections - Import a shell from an STL file using cross sections .....	91
Import IGES - Import an entire IGES file .....	94
Import ACIS - Import an entire ACIS file .....	96
Internal Volume - Calculates the internal volume of components or shapes.....	98
Length - Calculates the length of one or more components .....	99
Material Volume - Calculates the material volume of components or mesh shapes .....	100
Mesh Flowsplit Editor - Graphical editor to modify mesh flowsplit components.....	101
Model View Layout - Controls the layout of the model in the display window.....	102
Parallel Baffle to Baffle - Creates a baffle parallel to an existing baffle .....	103
Parallel Datum Plane - Creates a datum plane parallel to an existing datum plane.....	104
Polygon Vertices - Allows creation and editing of custom cross section shape by coordinate points	105
Print - Prints the graphical display.....	106
Print Preview - Previews the graphical display .....	107
Pipe Normal Plane Control Dialog – Location of a pipe normal plane .....	108
Set Rotation Point - Sets the anchor point for graphical rotations.....	109
Surface Area - Calculates the internal surface area of components or shapes.....	110
File → Options → General - Contains general options and preferences specific to the application...	111
File → Options → Favorites - Contains options regarding favorite folders and applications.....	112
File → Options → Save - Contains options and preferences regarding saving models .....	113
File → Options → Default Units - Contains the default units preferences .....	114
File → Options → Default Colors - Contains the default colors preferences.....	115
File → Options → Conversion - Contains the conversion preferences .....	116
File → Options → Discretization - Contains the discretization preferences .....	118
Translation - Translates a component.....	121
View Model Sectioning - Sections the model to view interior component .....	122
<b>CHAPTER 4: GEM3D Templates.....</b>	<b>124</b>
ActuatorConn3D - Actuator Connection .....	125
ControlLine - Control Line.....	127
ControlPoint - Control Point.....	128
CSBiRadial - Bi-Radial Cross Section .....	129
CSCircle - Circular Cross Section .....	130
CSCustom - User Defined Cross Section .....	131
CSEllipse - Elliptical Cross Section .....	132
CSRect - Rectangular Cross Section .....	133
CSRoundRect - Rounded Rectangular Cross Section .....	134
CSWire - Imported Cross Section .....	135
CustomDimension - User Created Custom Dimension .....	136
DatumPlane - Datum Plane .....	137
FESharedNodes - Node List for Imported Finite Element Mesh .....	139
GEMAssyConn - Assembly Connection .....	140
GEMBaffle - Baffle .....	141
GEMBaffleLeak - Leakage past a Baffle .....	144
GEMBPipe - Pipe with Bend.....	147
GEMCap - End Cap.....	155

GEMCatalystBrick - Flow-Through Catalyst Model .....	156
GEMChamber - Chamber of a Shell.....	165
GEMConnection - Connection .....	166
GEMConvectionConn – Convection between Thermal Mass and Volume .....	167
GEMConductanceConn – Conduction between Thermal Mass(es) and EngCylStrucCond .....	169
GEMCrankshaft – Crankshaft Builder .....	171
GEMFESharedNodes – Imported Finite Element Node List .....	174
GEMFESurfaceMesh – Imported Finite Element Surface Mesh .....	175
GEMFEVolumeMesh – Imported Finite Element Volume Mesh .....	176
GEMFlowDirection - Flow Port Direction.....	178
GEMFlowSystem – System of Multiple Pipes and Flowsplits.....	179
GEMFlowsplit - General Flowsplit .....	186
GEMFsplitGeneral - General Flowsplit.....	193
GEMInertia3D - Mechanical Rigid Body.....	200
GEMMechSolid3D – 3D Finite Element Solid Body.....	203
GEMMeshShape - General Mesh Shape .....	204
GEMMeshShell - Mesh Shell .....	205
GEMMiterBend – Miter Bend.....	213
GEMMultiplePipe - Bundle of Multiple Small Pipes.....	217
GEMOrifice - Orifice on the Surface of a Shell .....	224
GEMOrificeBaffle - Orifice through a Baffle .....	227
GEMOrificePipe - Internal Orifice in a Straight Pipe .....	231
GEMOrificePipeBend - Internal Orifice in a Bent Pipe .....	234
GEMOverPipe - Overlapping Pipes .....	237
GEMParticulateFilter - Wall-Flow Particulate Filter Model .....	244
GEMPerfAll - Perforation over Entire Component.....	253
GEMPerfCS - Perforated Section of a Baffle.....	257
GEMPerfPoint - Perforated Section on the surface of a Shell.....	261
GEMPerfRef - Perforated Section of a Straight Pipe, Shell, or Flowsplit.....	265
GEMPerfRefBend - Perforated Section of a Bent Pipe .....	269
GEMPipeXYZPoints - Pipe Formed Using X, Y, and Z Coordinates.....	273
GEMShell - ShellGEMShell - Shell .....	282
GEMSleeve - Sleeve on a Straight or XYZ Pipe.....	290
GEMSleeveBend - Sleeve on a Bent Pipe .....	296
GEMSolidFlowVol - Solid Flow Volume .....	302
GEMSolidShape - General Solid Shape .....	307
GEMSolidShell - Solid Shell.....	308
GEMSPipe - Straight Pipe .....	315
GEMSubAssExternalConn - External Subassembly Connection.....	323
GEMTank3D - Tank with 3-Dimensional Acceleration.....	324
GEMTank3DPort - Port for Tank3D .....	326
GEMThermalFE – 3D Finite Element Thermal Mass.....	327
GEMThermalFEPort – Surface Port on 3D FE Mesh .....	329
GEMThermalMass - Thermal Mass .....	331
GEMThermalMassPort – Surface Port of a Thermal Mass .....	333
GEMTsplit - T-Shaped Flowsplit .....	335

---

## Table of Contents

---

GEMVanePump – Detailed Vane Pump Model Template .....	342
GEMWoolAll - Wool Inside a Sleeve or Flowsplit.....	346
GEMWoolChamber - Wool Inside Shell Chamber .....	347
GEMWoolShell - Wool Inside Shell .....	348
GEMXsplit - X-Shaped Flowsplit .....	349
GEMYsplit - Y-Shaped Flowsplit .....	356
GerotorPump - Gerotor Pump Model Generator .....	364
LocalOrigin - Local Coordinate System.....	366
SensorConn3D - Sensor Connection .....	367
ThermalBound – Convection Boundary Condition .....	372
<b>CHAPTER 5: GEM3D Examples.....</b>	<b>373</b>
<b>INDEX.....</b>	<b>374</b>

## **CHAPTER 1: Introduction**

GEM3D is a tool that can be used to build 3D models of flow systems that can be discretized and made into model files for use with GT-SUITE. It provides the ability to build the model in a 3D environment so that the full details of the model can be included. It also includes sophisticated discretization logic that is able to transform the 3D model into a model file that is compatible with the GT-SUITE software.

GEM3D can be used to build any flow system that contains only flow components like pipes, mufflers, manifolds, air boxes, etc.

This manual describes the operations and components available in GEM3D. It is assumed that the user is already familiar with the GT-SUITE software package including GT-ISE and GT-POST. GEM3D will share concepts with both GT-ISE and GT-POST where possible to maintain consistency. It will also incorporate new concepts and methods that are unique to 3D modeling.

This manual documents the operations and components of GEM3D and can be a useful reference, but it is strongly recommended to attend a training class. Specific training classes on GEM3D may be available at various times, while general training classes on GT-SUITE are available on a regular basis. This approach is the fastest, most effective, and most enjoyable way to learn about GT-SUITE. Please see <https://www.gtisoft.com/events/trainings-and-seminars/> for information on training in the United States and Europe and <https://www.gtisoft.com/about-gti/contact-by-territory/> for contact information for our representatives in Japan, Korea, China, India, and Brazil.



## CHAPTER 2: GEM3D Menu Items

The menu items section contains descriptions for each toolbar menu item available in GEM3D. This also includes all operations and commands that may be available in GEM3D on right-click menus and shortcut keys. These descriptions can be found in the context help while using GEM3D by opening the help directly from the help menu. With the help open the menu items will be the first section available in the help tree.

### File Menu

The file menu has operations that deal with file handling and getting information into or out of the GEM3D application.



**Save:** Saves the active model file. If the current model file does not have a name, then this will act like the **Save As** command. [Keyboard hotkey = ctrl+S]



**Save As...:** Saves the active model file with a new name and/or location.



**Open:** Allows the user to open an existing model file. A GEM3D model file will have a .gem extension. [Keyboard hotkey = ctrl+O]



**Close:** Closes the active model file. [Keyboard hotkey = ctrl+F4]

**Recent:** Lists recent documents and directories for easy access. Single click a document to open it. Single click a directory to open the standard operating system file open dialog in that directory.

**Send:** This option allows for packaging up a model file to send via email, with a dedicated option to send the model directly to GT support engineers.

**Resources:** The default page for the File menu, includes sections for creating a new model file or library [Keyboard hotkey = ctrl+N], launching other GT-SUITE Applications or Utilities, or creating User Shortcuts. The majority of the space on the page is an HTML viewer, which may optionally be linked to any intranet or external URL.

**Examples:** Allows the user to quickly open GT provided example models.

**Tutorials:** Allows the user to quickly open GT provided tutorial documentation and models.

**Manuals:** Provides direct access to all User Manuals for GT-SUITE.

**Help:** This page offers various help options for the GEM3D application, including a link to open the Help file. [Keyboard hotkey = F1]

**Advanced:** Provides details about the current system and copyright information.



**Options :** Opens the application options window allowing modification of the local application properties. For information regarding the specific options available see the help for the [File > Options > General](#) dialog.







**Exit:** Exits the GEM3D application. [Keyboard hotkey = alt+F4]

## Home Menu

The home menu includes the most commonly needed operations for managing GEM models.



**New:** Creates a new model file. [Keyboard hotkey = ctrl+N]



**Open:** Allows the user to open an existing model file. A GEM3D model file will have a .gem extension. [Keyboard hotkey = ctrl+O]



**Save:** Saves the active model file. If the current model file does not have a name, then this will act like the **Save As** command. [Keyboard hotkey = ctrl+S]



**Save As...:** Saves the active model file with a new name and/or location.



**Import 3D:** Imports supported 3D geometry files (CAD). For more information regarding importing a CAD file, including supported file types, see the [Import 3D](#) command.



**Reload:** Reloads the active model file (similar to a refresh operation). This operation is used to refresh the display of the model in the graphical windows when a change or update is not shown. Due to the nature of 3D applications (like CAD) sometimes the graphical display of the model doesn't capture a change. When this happens, the reload operation should be done to verify if the problem is display related. This operation will fix a display related problem. If the problem persists after a reload operation, then the problem is not display related. [Keyboard hotkey = F5]



**Export ACIS...:** Exports the model as an ACIS file. This operation will open a dialog window allowing the name of the exported file to be given. This dialog will also allow the option to export the entire model as a single file or just the currently selected component. For more information regarding how to export files, see the [Export ACIS](#) command.



**Export STL...:** Exports the model as an STL file. This operation will open a dialog window allowing the name of the exported stl file to be given. This dialog will also allow the option to export the entire model as a single STL file or just the currently selected component. For more information regarding how to export stl files, see the [Export STL](#) command.



**Export Image...:** Exports the graphical display as an image. For more information regarding how to export an image, see the [Export Image](#) command. [Keyboard hotkey = ctrl+E]



**Close:** Close the active (selected) document. [Keyboard hotkey = ctrl+F4]

**Close All:** Close all documents that are open.





**Undo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row. [Keyboard hotkey = ctrl+Z]



**Redo:** Redoes the last operation. This operation can be used sequentially to redo multiple operations that were done in a row. [Keyboard hotkey = ctrl+Y]



**Stop:** Cancels the current operation. [Keyboard hotkey = Escape]



**Advanced Copy+Paste:** This operation will perform a normal copy operation along with a specialized paste operation. The specialized paste operation essentially involves pasting the copy of the components in a new location and/or orientation based on a transformation operation. There are 4 different transformation methods that can be done. [Keyboard hotkey = ctrl+T]

- **Translation:** This method allows the group of components to be translated in the X, Y, and/or Z directions (like doing a [Translation](#)). Choosing this option and clicking next will open a translation dialog window. This will allow the  $\Delta X$ ,  $\Delta Y$ , and  $\Delta Z$  distances to be specified. When finish is clicked, the group of components will be pasted and translated by the specified values.
- **Rotation:** This method allows the group of components to be rotated in an existing datum plane (like doing an [Assembly Rotation](#)). Choosing this option and clicking next will open a rotation dialog window and display the available datum planes in the graphical window. First, the reference datum plane in which the components will be rotated must be selected. Once the datum plane is selected, the angle field is enabled, allowing the angle of rotation to be specified. When finish is clicked, the group of components will be pasted and rotated by the specified angle.
- **Connection:** This method allows the group of components to be connected to another component (like doing a [Flow Connection](#)). Choosing this option and clicking next will display the available connection datum planes from the group of components and keep the dialog window open, even though it is not required anymore. First, the connection datum plane of the group of components must be selected. After this, all the available connection datum planes from other components will be displayed. Now, the connection datum plane must be chosen where the group of components is going to be connected to. Once this is selected, the group of components is copied and pasted such that the first selected datum plane is connected to the second selected datum plane.
- **Mirror:** This method option allows the group of components to be mirrored through a reference plane. Choosing this option and clicking next will open a mirror offset dialog window and display the available datum planes in the graphical window. First, the reference datum plane through which the components will be mirrored must be selected. Once the datum plane is selected, the offset field is enabled, allowing the offset distance to be specified. The offset distance specifies the distance the components will be placed away from a perfect mirror. An offset of 0 would result in the components being placed on the other side of the datum plane at the same distance away from the datum plane. A positive offset means they will be placed farther away from the plane than a direct mirror and a negative offset would place them closer. When finish is clicked, the group of components will be pasted and mirrored through the datum plane by the offset distance.





**Cut:** Performs a cut operation. [Keyboard hotkey = ctrl+X]



**Copy:** Performs a copy operation. [Keyboard hotkey = ctrl+C]



**Paste:** Performs a paste operation. [Keyboard hotkey = ctrl+V]

**Copy and Edit Object...:** Creates an identical copy of the selected component, gives it an incremental name, and opens it for editing. This is the same as making a copy of a component and then clicking the normal edit operation.



**Template Library:** Tiles the GEM model to the right and open the GT template library to the left.



**Find Template:** Search for the object name of a specific template in them model.



**Find Value:** Search for the value of a specific attribute in the model.



**Selection → General:** General selection mode enables different selection abilities depending on the current operation in GEM3D. Depending on the state of GEM3D (what operation is done), general selection mode will enable the selection of parts that can (and are allowed) be used in that operation. This avoids confusion by only allowing parts to be selected that can be used in the operation. Once the operation is finished, general selection will now allow selection of other parts.



**Selection → Quick Drill:** Quick drill selection mode is essentially a general selection mode that will skip any shell component if it is the first (outermost) component below the selection position (mouse click). This works just like [Drill Mode](#), but automatically selects the first component that would have been in the list.



**Selection → Chamber:** Chamber selection mode enables selection of individual chambers in shells.



**Selection → Body:** Body selection mode enables selection of components, features, and datum planes.



**Selection → Control Elements:** Control elements selection mode enables selection of control points, control lines, datum planes, and baffles. This mode is very useful when creating dimensions since it allows selection of multiple elements that are useful while dimensioning while not selecting the components that may get in the way.



**Selection → Vertex:** Vertex selection mode enables selection of control points.



**Selection → Line:** Line selection mode enables selection of control lines.





**Box Selection:** Box selection mode enables selection of multiple components and shapes by dragging a 2D box around the desired components and shapes. Upon entering box selection mode, the perspective will remain locked and the user can drag a box around multiple parts. This selection box includes anything that it touches; the component does not need to be fully contained within the selection box to be selected.



**Drill Mode:** Drill mode allows selection of components and features that are inside or below other components. This works in conjunction with the current selection mode. When selected, drill mode makes any selection with the mouse return a list of all components or features that were intersected with the mouse click location. The particular component or feature can then be selected or edited from this list. Clicking on the component or feature name will highlight the part in the graphical window. Selecting this operation again will exit drill mode. *[Alt + left mouse button] on Windows PC. [d + left mouse button] on Linux.*



**Unselect All:** This operation will unselect all components and features that are selected. This operation can also be done by double-clicking with the left mouse button in the graphical window where no part of the model is (background).



**Case Setup...:** Opens Case Setup. For more information regarding how to use Case Setup, see the [Case Setup](#) command. [Keyboard hotkey = F4]



**Export GT Model...:** Discretizes the active model and exports a model file (.gtsb or .gtm) for use in GT-ISE. This operation will open a dialog window with the required input to discretize the model and create the model file. For more information regarding how to export the model file, see the [Export Model](#) command.

## View Menu

The view menu includes operations related to the visual display of the model and graphical operations.



**Add Note:** Notes can be added to the canvas to serve as reminders and to aid future users who might inherit a model.



**Model Sectioning...** : Opens the model sectioning dialog window which will create a temporary cutting plane on the model allowing the graphical display to show the inside of the model. This can be used to look inside of shells to see internal components and to create screen shots for presentations. For more information regarding model sectioning, see the [Model Sectioning](#) command.



**Home View:** Resets the graphical window by rotating the display to the top view and zooming to the default level (1:1). [Keyboard hotkey = F4]



**Orientation:** Contains the display rotation options.



**Isometric (XYZ):** Rotates the display to show the isometric view of the model.





**Front View (YZ):** Rotates the display to show the front view of the model. The front view is defined as the view looking at the YZ plane.



**Back View (-YZ):** Rotates the display to show the back view of the model. The back view is defined as the view looking at the -YZ plane.



**Top View (XY):** Rotates the display to show the top view of the model. The top view is defined as the view looking at the XY plane.



**Bottom View (-XY):** Rotates the display to show the bottom view of the model. The bottom view is defined as the view looking at the -XY plane.



**Right View (XZ):** Rotates the display to show the right view of the model. The right view is defined as the view looking at the XZ plane.



**Left View (-XZ):** Rotates the display to show the left view of the model. The left view is defined as the view looking at the -XZ plane.



**Zoom In:** Zooms in on the model. Scrolling up with the mouse wheel will also zoom in. [Keyboard hotkey = NumPad +]



**Zoom Out:** Zooms out from the model. Scrolling down with the mouse wheel will also zoom out. [Keyboard hotkey = NumPad -]



**Zoom 1:1:** Restores the zoom to the default level (1:1). [Keyboard hotkey = Equals]



**Zoom Section:** Zooms to a user specified section defined by a box created by clicking and dragging the mouse.



**Axis:** Option to show and hide the global coordinate axis. The global coordinate system (also called the world coordinate system) represents the coordinates by which components are located in space. The origin of the global coordinate system is always displayed as a small black dot in the graphical display.



**Lower Left Corner:** This option will display the global coordinate system axis in the lower left corner of the graphical display. This display is not part of the model and will always remain in the lower left corner no matter what the orientation of the model is.



**Origin:** This option will display the global coordinate system axis at the origin. This display is part of the model and will remain at the origin when the model orientation is changed.



**Hide Axis:** This option will hide the axis.





**Render:** Option to change how the model is rendered (drawn) in the graphical window. This does not affect the model geometry or resolution, only the way the model is drawn graphically on the canvas.

**Solid:** Displays all surfaces as continuous.

**Wireframe:** Displays all surfaces using lines to show the sides of each triangle used to draw the model.



**Perspective:** Toggle to switch to a perspective view. By default the view is set to orthographic. An orthographic view assumes two vanishing points located at infinity so all lines appear parallel. If this menu item is checked, then a perspective view will be shown. A perspective view has the vanishing points located a finite distance away from the model so it will look more physical. Un-checking this menu item will return to an orthographic view. This option will be saved as a local application property.



**Set Rotation Point:** Opens the set rotation point dialog window. This is used to set the anchor point for graphical rotation operations. For more information regarding set rotation point, see the [Set Rotation Point](#) command.



**Edit Objects in Spreadsheet:** The table edit view will show a list of attributes for each template type in the graphical window. This view type can be used to compare the values of attributes in the same template type. It can also be used to easily check or edit the attributes of many components. This is equivalent to the table edit view in GT-ISE.



**Connection Datums:** Toggle to show or hide the connection datum planes for a single component.



**Draft Mode:** Toggle to switch between normal and draft mode. Normal mode will display all perforate holes as actual holes through the parent component or feature. This makes the model look good, but it will make display operations slower. Draft mode will display all perforate holes as simple circles on the surface of the parent component or feature. Draft mode makes display operation very fast. Draft mode can be used while building a model to maintain fast program operation. Normal mode can be used to see how the physical model would appear and for presentations and demonstrations. The default option is draft mode.



**Drilled Holes:** Toggle to show all drilled holes in the model. The drilled holes are holes in components or features that are created due to intersection with other components or features. By default these holes are not drawn because they will slow down graphical operations. Checking this menu item will redraw the model with the holes show for a more physically accurate model. This can be used to show the model or create presentations. Because of the performance reduction, the show drilled holes mode will put the GEM3D model in a restrictive state where only graphical operations are allowed. To edit the model this menu item must be un-checked to exit show drilled holes mode. The status bar at the bottom of the application will show the current status of the model.





**Model View Layout...** : Opens the model view layout dialog window. This is used to change the number of views, what each view shows, and the layout of the views in the graphical window. For more information regarding model view layout, see the [Model View Layout](#) command.



**Arrange Windows** : Controls the arrangement and layout of all open windows.



**Cascade**: Stacks all open windows in a diagonal layout.



**Arrange Windows...**: Opens the arrange windows dialog window allowing full control over all open windows.



**Tile Horizontally**: Tiles all windows horizontally.



**Tile Vertically**: Tiles all windows vertically.



**Customize Toolboxes**: Opens the customize toolbox window which allows customized toolboxes to be created that contain any of the operations that are available in GEM3D. These toolboxes will be saved with the application.

## Flow Menu

The flow menu includes operations specific to building or converting fluid flow systems.



**Straight Pipe...** : Creates a straight pipe that can have any cross sectional shape. For more information see the help for the '[GEMSPipe](#)' template.



**Bend Pipe...** : Creates a bent pipe. For more information see the help for the '[GEMBPipe](#)' template.



**TSplit...** : Creates a flowsplit that is shaped like a T. For more information see the help for the '[GEMTSplit](#)' template.



**YSplit...**: Creates a flowsplit that is shaped like a Y. For more information see the help for the '[GEMYsplit](#)' template.



**Shell...** : Creates a shell that will be discretized into smaller volumes. For more information see the help for the '[GEMShell](#)' template.



**Pipe XYZ Points...** : Creates a system of pipes including straight and bent sections. For more information see the help for the '[GEMPipeXYZPoints](#)' template.





**Aftertreatment** : Contains aftertreatment components. Available components are listed below. Clicking the arrow expands the selection to allow choice of any component. Clicking the icon directly will create the first component in the list.



**Catalyst...** : Creates a catalyst brick. For more information see the help for the '[GEMCatalystBrick](#)' template.



**Particulate Filter...** : Creates a particulate filter (PF) that can represent a diesel (DPF) or gasoline (GPF) application. For more information see the help for the '[GEMParticulateFilter](#)' template.



**Advanced:** Contains advanced, more specialized flow components. Available components are listed below. Clicking the icon or arrow expands the selection to allow choice of any component.



**Multiple Pipe...** : Creates a bundle of pipes. For more information see the help for the '[GEMMultiplePipe](#)' template.



**Overlapping Pipe...** : Creates an overlapping pipe segment. For more information see the help for the '[GEMOverPipe](#)' template.



**Flowsplit...** : Creates a general flowsplit that can have any shape. For more information see the help for the '[GEMFlowsplit](#)' template.



**XSplit...** : Creates an X-shaped crossover pipe. For more information see the help for the '[GEMXsplit](#)' template.



**Pump:** Contains pump components. Available components are listed below. Clicking the icon or arrow expands the selection to allow choice of any component.



**Gerotor Pump...** : Creates a gerotor type pump. For more information see the help for the '[GerotorPump](#)' template.



**Vane Pump...** : Creates a vane type pump. For more information see the help for the '[GEMVanePump](#)' template.



**Flow Connection:** Creates a flow connection between 2 components by moving the first component to the second. Clicking this option will switch to flow connection mode and the mouse cursor will change. This will also automatically display all available connection datum planes for each component. To make the connection, first click on the connection datum plane of the component that needs to be moved. Then, click on the connection datum plane of the fixed component. This will create a flow connection by moving the first component to the second component such that the selected connection datum planes are coplanar. Flow connection mode will still be enabled until it is canceled (Escape key). The status bar at the bottom of the application





will specify what needs to be done for the current step during the operation. This is the standard connection used to connect flow components together to make a system.



**Component Flow Connection:** Creates a flow connection between 2 components by moving the first component to the second. This works in the same way as flow connection except that the actual component is selected instead of the connection datum plane. Since the component is selected, GEM3D will determine which face of the first component to move to which face of the second component based on relative distance. The 2 closest available faces will be connected together. This provides no added functionality over flow connection, but may offer an easier interface.



**Extruded Connection:** Creates an extruded connection between 2 components by creating a small section of pipe between the components. This works in the same way as flow connection except that the second component is not moved. A small section of pipe is added to graphically represent the connection between the components. This small section of pipe will not be modeled during discretization, so an extruded connection is restricted to very short distances. Extruded connection mode will still be enabled until it is canceled (Escape key). The status bar at the bottom of the application will specify what needs to be done for the current step during the operation. An extruded connection should typically be used when 2 components need to be connected, but both of them are positioned such that neither can or should be moved.



**Assembly Connection:** Creates an assembly connection between 2 or more components. The assembly connection creates a connection between multiple components without moving them. This works like a group operation. When any one of the components of an assembly is moved, the other(s) will move along with it to maintain their relative location. To create an assembly connect simply select any components that should be included in the connection. Then click this option and an assembly connection will be created between all the selected components. This connection is typically used to connect a muffler shell with its internal pipes.



**Boundary Manager:** Opens the boundary manager dialog window. The boundary manager allows management of all boundary ports of the model to control names, ID numbers, and flow directions for all boundaries of the model. More information can be found in the help for the [Boundary Manager](#) dialog window.



**Add Connection...:** Adds a connection to a port of the selected component. This will open the port connection dialog window that will allow the selection of which port to add the connection to and which type of connection to add. This should be used to add a non-default orifice connection or a different type of connection to a flow component. A default orifice connection will always be placed between flow components during discretization. Therefore, this operation is only required when a default orifice connection is not appropriate.



**Disconnect Components:** Deletes the connection between the two selected components without moving either of the components.



**Move Component:** Moves a selected component to a second component by making their respective connection datum planes coplanar. Clicking this command will switch to move component mode and the mouse cursor will change. This will also automatically display all available connection datum planes for each component. To move a component, first click on the connection datum plane



of the component that needs to be moved. Then, click on the connection datum plane of the fixed component. This will move the first component to the second component such that the selected connection datum planes are coplanar. Move component mode will still be enabled until it is canceled (Escape key). The status bar at the bottom of the application will specify what needs to be done for the current step during the operation.



**Datum Planes:** Contains functions for adding a '[DatumPlane](#)' to the model, either a **Global Datum Plane** (exists independently in the model tree) or as a **Child Datum Plane** (belongs to a specific component). The different methods available for creating datum planes are identical for global and for child planes, and are therefore described only once below. Note that a specific component must be selected in the model for the Child Datum Plane options to be enabled in the toolbar.



**Pipe Normal :** Allows a local datum plane to be placed on the model that is perpendicular (normal) to the estimated flow direction in that section. This is a very specific operation that is designed for pipe-like shapes. When this operation is clicked, a point on a mesh or solid shape may be selected. After this point is selected, a datum plane will be placed perpendicular to the estimated flow direction on the shape and the [Pipe Normal Plane Control dialog](#) will open. The plane will be shown on the shape as a transparent green plane. After selecting the plane location, the '[DatumPlane](#)' object editor dialog will open. Pressing the Esc key or the [cancel operation](#) button at any time will cancel the creation of the plane.



**Snap to Feature ...:** Allows a datum plane to be positioned by clicking when the displayed preview of the plane snaps to the desired geometric feature of the shape nearest the mouse cursor. After selecting the plane location, the '[DatumPlane](#)' object editor dialog will open. This approach works best on solid shapes and on flat surfaces of Mesh Shapes.



**3 Points ...:** Allows a datum plane to be placed on the model based on the selection of 3 points to define the plane. When this operation is clicked, the mouse cursor will change to a cross and any point on a mesh shape can be selected. 3 points must be selected. These 3 points will then uniquely define a plane to be used as the datum plane. After the third point is selected, the '[DatumPlane](#)' object editor dialog will open.



**Single Point and Vector ...:** Allows a datum plane to be placed on the model based on manual definition of a point in 3D space {X, Y, Z} and a vector direction. This option directly opens the '[DatumPlane](#)' object editor dialog (no graphical interaction in 3D canvas).



**Parallel Datum Plane:** Creates a datum plane parallel to an existing datum plane. To do this first select an existing datum plane. Then click on this command and the parallel datum plane dialog window will open. In this window the distance away from the existing datum plane must be specified. Clicking OK will open a '[DatumPlane](#)' template that has attributes pre-filled such that the datum plane will be created parallel to the selected datum plane. Clicking OK or Apply will create the new datum plane. For more information see the help for the '[DatumPlane](#)' template.





**Perpendicular Datum plane:** Creates a datum plane perpendicular to an existing datum plane. To do this first select an existing datum plane. Then click on this command and a '[DatumPlane](#)' template will open that has attributes pre-filled such that the datum plane will be created perpendicular to the selected datum plane. Clicking OK or Apply will create the new datum plane. For more information see the help for the '[DatumPlane](#)' template.



**Circle:** Creates a circular cross section.



**Rectangle:** Creates a rectangle shaped cross section.



**Round Rectangle:** Creates a rectangle shaped cross section with rounded corners (fillets).



**Ellipse:** Creates an elliptical shaped cross section.



**BiRadial:** Creates a bi-radial cross section consisting of 2 different radius sections for the top/bottom and sides.



**Custom:** Creates a custom cross section that can be any shape.

## Thermal Menu

The thermal menu includes operations specific to building or converting thermal systems.



**EngCylStrucCond:** Allows the building of a parametric cylinder structure component in GEM3D. This object is used to model the cylinder structure of an engine using a finite element representation of the cylinder liner, piston, and head (including ports, valves and valve guides not shown in the picture below). This object should be used when the cylinder structure will be connected to coolant and/or oil flow circuits. In GEM3D, connections can be made from this component to neighboring [GEMThermalMass](#) or [GEMSolidFlowVol](#) components through a conduction or convection connection.



**Thermal Mass Port:** This action creates a surface on a [GEMThermalMass](#) component. This surface can have the boundary conditions provided directly in GEM3D, or they can be linked in GT-ISE.



**Thermal FE Port:** This action creates a surface on a [GEMThermalFEPort](#) component. This surface must have the boundary conditions linked to it in GT-ISE.



**Datum Planes:** Contains functions for adding a '[DatumPlane](#)' to the model, either a **Global Datum Plane** (exists independently in the model tree) or as a **Child Datum Plane** (belongs to a specific component). The different methods available for creating datum planes are identical for global and for child planes, and are therefore described only once below. Note that a specific component must be selected in the model for the Child Datum Plane options to be enabled in the toolbar.





**Pipe Normal** : Allows a local datum plane to be placed on the model that is perpendicular (normal) to the estimated flow direction in that section. This is a very specific operation that is designed for pipe-like shapes. When this operation is clicked, a point on a mesh or solid shape may be selected. After this point is selected, a datum plane will be placed perpendicular to the estimated flow direction on the shape and the [Pipe Normal Plane Control dialog](#) will open. The plane will be shown on the shape as a transparent green plane. After selecting the plane location, the 'DatumPlane' object editor dialog will open. Pressing the Esc key or the [cancel operation](#) button at any time will cancel the creation of the plane.



**Snap to Feature ...**: Allows a datum plane to be positioned by clicking when the displayed preview of the plane snaps to the desired geometric feature of the shape nearest the mouse cursor. After selecting the plane location, the 'DatumPlane' object editor dialog will open. This approach works best on solid shapes and on flat surfaces of Mesh Shapes.



**3 Points ...**: Allows a datum plane to be placed on the model based on the selection of 3 points to define the plane. When this operation is clicked, the mouse cursor will change to a cross and any point on a mesh shape can be selected. 3 points must be selected. These 3 points will then uniquely define a plane to be used as the datum plane. After the third point is selected, the 'DatumPlane' object editor dialog will open.



**Single Point and Vector ...**: Allows a datum plane to be placed on the model based on manual definition of a point in 3D space {X, Y, Z} and a vector direction. This option directly opens the 'DatumPlane' object editor dialog (no graphical interaction in 3D canvas).



**Perpendicular Datum plane**: Creates a datum plane perpendicular to an existing datum plane. To do this first select an existing datum plane. Then click on this command and a 'DatumPlane' template will open that has attributes pre-filled such that the datum plane will be created perpendicular to the selected datum plane. Clicking OK or Apply will create the new datum plane. For more information see the help for the 'DatumPlane' template.



**Parallel Datum Plane**: Creates a datum plane parallel to an existing datum plane. To do this first select an existing datum plane. Then click on this command and the parallel datum plane dialog window will open. In this window the distance away from the existing datum plane must be specified. Clicking OK will open a 'DatumPlane' template that has attributes pre-filled such that the datum plane will be created parallel to the selected datum plane. Clicking OK or Apply will create the new datum plane. For more information see the help for the 'DatumPlane' template.



**Convection Connection**: This action creates a convection connection between a [GEMThermalMass](#) and a [GEMSolidFlowVol](#), [Straight Pipe](#), [Pipe in 3D with Bends](#), or [Bent Pipe](#). The specific convection surface can be defined along with the heat transfer coefficient, those values will be used when the GT-ISE file is created to make the convection connection.





**Conduction Connection:** This action creates a conduction connection between a [Thermal Mass](#) component and either another thermal mass component or an EngCylStrucCond. Connections between adjacent EngCylStrucCond components should be made in GT-ISE using the FEStructureConn connection instead.



**Conduction Connections Wizard:** This action will analyze the Thermal Mass components in the model and determine if a conduction connection can be created between them. When activated, a list will appear with the potential conduction connections to be made, individual connections can then be selected or removed. This wizard will automatically run when the GT-ISE file is created so that conduction connections can be added between neighboring components.

## Mechanical Menu

The mechanical menu includes operations specific to building or converting mechanical systems.



**Crank Shaft:** This component creates a crankshaft from imported data. The solid shape should be imported and cut into the various components (journals, webs, pins, flywheels, shafts, split pin transfer plates). The cut components are then named in the GEMCrankShaft part.



**Flexible Body Surface Port:** Creates a surface port on the Mechanical Finite Element part for contact with another component. The connections to this surface must be made in GT-ISE.



**Flexible Body Node Port:** Creates a single (or multiple) node port on the Mechanical Finite Element part. Nodes selected by this action can either have a contact with another component or can be used for plotting.



**Datum Planes:** Contains functions for adding a '[DatumPlane](#)' to the model, either a **Global Datum Plane** (exists independently in the model tree) or as a **Child Datum Plane** (belongs to a specific component). The different methods available for creating datum planes are identical for global and for child planes, and are therefore described only once below. Note that a specific component must be selected in the model for the Child Datum Plane options to be enabled in the toolbar.



**Pipe Normal :** Allows a local datum plane to be placed on the model that is perpendicular (normal) to the estimated flow direction in that section. This is a very specific operation that is designed for pipe-like shapes. When this operation is clicked, a point on a mesh or solid shape may be selected. After this point is selected, a datum plane will be placed perpendicular to the estimated flow direction on the shape and the [Pipe Normal Plane Control dialog](#) will open. The plane will be shown on the shape as a transparent green plane. After selecting the plane location, the '[DatumPlane](#)' object editor dialog will open. Pressing the Esc key or the [cancel operation](#) button at any time will cancel the creation of the plane.





**Snap to Feature ...:** Allows a datum plane to be positioned by clicking when the displayed preview of the plane snaps to the desired geometric feature of the shape nearest the mouse cursor. After selecting the plane location, the **'DatumPlane'** object editor dialog will open. This approach works best on solid shapes and on flat surfaces of Mesh Shapes.



**3 Points ...:** Allows a datum plane to be placed on the model based on the selection of 3 points to define the plane. When this operation is clicked, the mouse cursor will change to a cross and any point on a mesh shape can be selected. 3 points must be selected. These 3 points will then uniquely define a plane to be used as the datum plane. After the third point is selected, the **'DatumPlane'** object editor dialog will open.



**Single Point and Vector ...:** Allows a datum plane to be placed on the model based on manual definition of a point in 3D space {X, Y, Z} and a vector direction. This option directly opens the **'DatumPlane'** object editor dialog (no graphical interaction in 3D canvas).



**Perpendicular Datum plane:** Creates a datum plane perpendicular to an existing datum plane. To do this first select an existing datum plane. Then click on this command and a **'DatumPlane'** template will open that has attributes pre-filled such that the datum plane will be created perpendicular to the selected datum plane. Clicking OK or Apply will create the new datum plane. For more information see the help for the **'DatumPlane'** template.



**Parallel Datum Plane:** Creates a datum plane parallel to an existing datum plane. To do this first select an existing datum plane. Then click on this command and the parallel datum plane dialog window will open. In this window the distance away from the existing datum plane must be specified. Clicking OK will open a **'DatumPlane'** template that has attributes pre-filled such that the datum plane will be created parallel to the selected datum plane. Clicking OK or Apply will create the new datum plane. For more information see the help for the **'DatumPlane'** template.

## Convert Menu

The convert menu includes operations for converting imported 3D geometry into GT specific flow, thermal, or mechanical components.



**Mark Surface:** Allows sections of mesh shapes to be marked (painted). When this operation is clicked, the mouse cursor will change to denote the marking mode and the mark tolerance dialog window will open. Clicking on a section of a mesh shape will mark the surface according to the edge detection angle specified in the mark tolerance window. This angle represents the detection angle between adjacent triangles that will cause a mark operation to stop. An angle of 0° represents a very strict criteria where adjacent surfaces have to be exactly parallel (0 angle between them) to be marked. An angle of 90° represents full freedom where adjacent surfaces can be at any angle, essentially marking the entire mesh shape. The detection angle can be changed using the slider in the window. There are also 3 existing levels described below that can be set by the buttons on the left side of the window. The mark operation can be canceled by pressing the Esc key, the [cancel](#)





[operation](#) button, closing the mark tolerance dialog with the cancel or X buttons, or clicking on the mark operation button.



**Mark Triangle:** Sets the edge detection angle to 0°. This will make each click mark a single triangle of the mesh shape.



**Mark Face:** Sets the edge detection angle to 5°. This will make each click mark an entire face of the mesh shape. This is typically used to mark flat surfaces on a mesh shape that need to be removed. A common example may be caps on the end of flow paths.



**Mark Surface:** Sets the edge detection angle to 35°. This will make each click mark an entire surface of the mesh shape. This is typically used to mark continuous surface sections on a mesh shape. This may typically be used to mark large smooth sections that have a sharp discontinuity to an adjacent surface. This option may provide an easier method than the cutting plane in some circumstances. A common example may be an inlet pipe that is flush where it enters a large volume.



**Undo Marking:** Undoes the last mark operation. This operation can be used sequentially to undo multiple mark operations that were done in a row.



**Redo Marking:** Redoes the last mark operation. This operation can be used sequentially to redo multiple mark operations that were done in a row.



**Delete Surface:** Deletes any currently painted sections in the model. This works in a similar way to the clip operation, except this operation deletes the section after separating it from the original mesh shape.



**Pipe Normal Cutting Plane:** Allows a local cutting plane to be placed on the model that is perpendicular (normal) to the estimated flow direction in that section. This is a very specific operation that is designed to cut pipe-like shapes. When this operation is clicked, a point on a mesh or solid shape may be selected. After this point is selected, a local cutting plane will be placed perpendicular to the estimated flow direction in the mesh shape and the [Pipe Normal Plane Control dialog](#) will open. The cutting plane will be shown on the shape as a transparent green plane. Pressing the Esc key or the [cancel operation](#) button at any time will cancel the creation of the local cutting plane.

- Clicking on the plane and dragging the mouse will translate the local cutting plane along the flow path of the particular mesh shape. As the plane is dragged, it will maintain its perpendicular orientation based on the current position of the plane.
- All other graphical operations (rotation, translation, zoom, etc.) can be done with a local cutting plane on the model.



**Snap to Feature Cutting Plane:** Allows a cutting plane to be positioned by clicking when the displayed preview of the plane snaps to the desired geometric feature of the shape nearest the mouse cursor. After selecting the plane location, the [Cutting Plane Control Window](#) will open. This approach works best on solid shapes and on flat surfaces of Mesh Shapes.





**3 Points Cutting Plane:** Allows a cutting plane to be placed on the model based on the selection of 3 points to define the plane. When this operation is clicked, the mouse cursor will change to a cross and any point on a mesh shape can be selected. 3 points must be selected. These 3 points will then uniquely define a plane to be used as the cutting plane. After the third point is selected, the cutting plane will be shown on the model along with an arrow normal to the plane and the [Cutting Plane Control Window](#) will open. Pressing the Esc key or the [cancel operation](#) button at any time will cancel the creation of the cutting plane.

- Clicking on the plane and dragging the mouse will translate the cutting plane.
- Clicking on the tips of the arrow and dragging the mouse will rotate the cutting plane.
- Clicking on the base of the arrow (the ball at the intersection with the plane) and dragging the mouse changes the center of rotation.
- All other graphical operations (rotation, translation, zoom, etc.) can be done with a cutting plane on the model.



**Restore Cutting Plane:** Places a cutting plane in the same location and orientation as the last cutting plane that was used. This cutting plane is exactly like the operation above and can be rotated and translating into position. This is intended to provide an easy way to create a cutting plane based on the last location so that only simple rotations and/or translations can be used to position this plane for the next cut.



**Separate by Curves:** This operation will separate a mesh shape into many mesh shapes, and can be used on single or multiple meshes. To separate the mesh shapes GEM3D identifies seams in the mesh where adjacent triangles do not share the same vertex points. At areas where this occurs, the triangles will be assigned to different mesh shapes, with neighboring triangles which share vertex points being grouped to the same mesh.

A "tolerance" slider is available to adjust the sensitivity of the operation; increasing the tolerance value will generally create more separate meshes. The OK button closes the window, using the last applied tolerance or using the default tolerance if a new tolerance has not been applied. The Cancel button reverses all applied changes (keeping the original mesh together) and closes the window. The Preview button uses the current tolerance to separate the selected mesh, and keeps the window open so changes can be made to the tolerance value.

Separate by Curves is a very powerful option to automatically divide large imported files. This operation should be used whenever an imported mesh shape consists of several parts. If an appropriate tolerance value cannot be found for a given mesh, then the mesh can be manually separated using the Cutting Plane, Local Cutting Plane, and Marking operations.



**Clip Shape:** Clips a mesh shape(s) into multiple mesh shapes. This operation should be used in conjunction with the local cutting plane, cutting plane, or mark operations. Before this operation is enabled a cutting plane must be present on the model or at least 1 section must be marked. If marked sections exist, then clicking this operation will separate the marked sections from the original mesh shape and make new mesh shapes out of them. If a cutting plane exists, then 1 or more mesh shapes must be selected to enable the clip operation. Then, clicking this operation will cut all selected mesh shapes at the cutting plane and make new mesh shapes from the sections. If a local cutting plane exists, then clicking this operation will cut only the mesh shape that the local cutting plane was created on.







**Merge Meshes:** This operation will merge (combine) multiple mesh shapes into a single mesh shape. To enable this operation each mesh shape to be merged must first be selected. Once multiple mesh shapes are selected, this operation will merge them together. In order for mesh shapes to be merged, they must share a common boundary. If they do not, then they cannot be merged together.



**Select All Shapes:** This operation selects all shapes (mesh and solid shapes) in the model. This may typically be used after placing a cutting plane so that all shapes are included in a clip operation.



**Filter Shapes:** Passes all the shapes (mesh and solid shapes) in the model through a filter to help determine which ones are actual components. For more information see the help for the [Filter Shapes](#) dialog.



**Patch Mesh Ports:** This operation will physically patch open ports of a mesh shape. Any port of a mesh shape that is not to be used should be patched to provide a better conversion. The ports do not have to be patched, but if the conversion fails, then patching the unused ports may solve the problem and allow the mesh shape to be converted. To start the patching operation, click this button. This will identify the open ports of all mesh shapes. To patch a port, just click on it with the mouse button. Pressing this button again, the Esc key, or the [cancel operation](#) button at any time will cancel the patching operation.



**Convert Shape to Component:** This operation is used to convert mesh shapes ('[GEMMeshShape](#)') and solid shapes ('[GEMSolidShape](#)') into GEM3D components. Shapes cannot be discretized by GEM3D, and thus all shapes that are required in the discretized model must first be converted into GEM3D components. More information about converting shapes can be found in the help for the [Convert Shape to Component](#) dialog.



**De-convert Component:** This operation will convert an existing GEM3D component back into the original mesh shape ('[GEMMeshShape](#)') or solid shape ('[GEMSolidShape](#)') that it was created from. This operation is only available if the selected component was created from a shape and the "Allow Mesh Deconversion" option is checked in [File>Options>Conversion](#). A GEM3D component that was created from a shape can be identified by a small black diamond in the upper left hand corner of the icon in the tree.



**Remove Deconversion Data:** This operation will remove the original mesh data that is required for a deconversion operation (see above). This would operate on any GEM components that were converted from imported shapes ('[GEMMeshShape](#)' or '[GEMSolidShape](#)') and have deconversion data available. The operation can remove the data from a single component, a selected subset, or all components. Since the mesh data must be maintained and handled, this will increase both memory usage and computational speed. If the response of the application is slow, then removing the deconversion data will potentially improve the speed and memory usage. If the response is normal, then there is no need to remove the deconversion data.



**Convert to Mesh Shape:** This operation is used to convert imported solid shapes or FE surface meshes into mesh shapes. This enables the resulting mesh shape to then be converted into various



GT flow components (pipes, shell, etc.). Note that imported solid shapes can often be converted directly to GT flow components so this operation is not always necessary. When FE surface meshes are converted to mesh shapes, an option is given to save the original shape in the model tree (copy and convert). This option is required when the purpose is to perform a thermal co-simulation or to export boundary conditions for an external FEA tool. In this case, maintaining the original FE mesh in the tree after conversion allows for a later operation to “map” the FE mesh elements to the converted GT flow volumes.

## Dimension Menu

The dimension menu includes functions for measuring or adding dimensions to models.



**Measure Distance:** Measures the distance between any two points in the model. When measuring distance, any 2 points in the model may be clicked and this operation will determine the straight line distance between them.



**Measure Diameter:** Measures the diameter on any recognized closed cross section in the model. This will also return the area and perimeter as well. This can be used on any shapes ('GEMMeshShape' or 'GEMSolidShape') if they have well defined port openings.



**Internal Volume:** Calculates the internal volume of the currently selected component(s) or shape(s) and opens the volume calculation dialog window. For more information see the help for the [Internal Volume](#) dialog window.



**Automatic Mode:** Shows the automatic dimensions for components and features. Currently this is only supported for particular components and features. To see the automatic dimensions simply select the component or feature and then click this option. To see the automatic dimensions for other components or features, simply select them with the mouse. To stop showing the automatic dimensions, simply click this option again.



**Length:** Measures the length of a component, or a selected group of components, along the flow path. The length of the individual parts is calculated, as well as the total length of the selection. For more information see the help for the [Length](#) dialog window.



**Show Porosity:** Calculates the porosity of any currently selected perforate sections and opens the porosity calculation dialog window. This operation can also be done when a component is selected that has one or more perforate features. This will result in the porosity of each perforate section being included in the calculation. The porosity of a perforate section is defined as the open area of all holes divided by the surface area that the perforate section occupies. Since the inside geometries are important for the GT-SUITE flow simulation, all surface areas used are for the inside surfaces.



**Surface Area:** Calculates the surface area of the currently selected component(s) or shape(s) and opens the surface area calculation dialog window. For more information see the help for the [Surface Area](#) dialog window.





**Material Volume:** Calculates the material volume of the currently selected component(s) or shape(s) and opens the volume calculation dialog window. For more information see the help for the [Material Volume](#) dialog window.



**Center of Gravity:** Calculates the center of gravity of the currently selected component(s) and opens the center of gravity dialog window. For more information see the help for the [Center of Gravity](#) dialog window.



**Display Control Points:** Toggle to show or hide control points for a single component or for a group of components. Since this operation can operate on multiple components, the actual operation depends on the current selection.

- If a single component is selected, it will display the control points for the single component. If that component's control points are already displayed, then this will hide the control points.
- If no component is selected, then this option would display the control points for all components in the model. If ANY components already have control points displayed, this option would hide ALL control points.
- If multiple components are selected, this option would show all control points for all selected components (and vice versa). If ANY of the selected components already have control points shown, then this would hide ALL control points.



**Distance Dimension:** Creates a distance dimension between 2 selected elements. The distance dimension that is created will depend on which types of elements are selected. Refer to the table below for what distance dimension will be created for each selection pair. For more information see the help for the [CustomDimension](#) template.

Distance Dimensions	Components	Datum planes	Control Points	Lines
Components	Straight distance from center point of each origin.	Straight distance from component center point origin to center point of datum plane.	Straight distance from component center point origin to control point.	Straight distance from component center point origin to center point of line.
Datum planes		Normal distance between parallel planes.	Normal distance.	Normal distance between parallel plane and line.
Control Points			Straight line distance.	Normal distance between line and point.
Lines				Normal distance between parallel lines.



**Angle Dimension:** Creates an angle dimension using 3 control points or 2 control lines. For more information see the help for the [CustomDimension](#) template.





**Create Control Point:** Contains operations to create control points. Available operations are listed below. Clicking the arrow expands the selection to allow choice of any operation. Clicking the name directly will start the first operation in the list.



**Create Component Control Point:** Creates a control point at the location of the mouse click. Clicking this option will switch to control point creation and change the mouse cursor icon. Clicking any location on a component, feature, or shape will create a control point. The resulting control point will belong to the component (child-parent relationship) and move accordingly when the component is modified. Its location is determined by the parent component, and therefore the location cannot be edited. For more information see the help for the '[ControlPoint](#)' template.



**Create Global Control Point...:** Creates a control point at a location specified by an X, Y, and Z coordinate. Clicking this option will open a control point dialog window which allows the entry of an X, Y, and Z coordinate directly. The resulting control point will not belong to any component and can be edited by directly changing its location. For more information see the help for the '[ControlPoint](#)' template.



**Convert to Global Control Point:** Converts a component control point to a global control point. This operation is allowed when any component control point is selected.



**Create Control Line:** Creates a control line between 2 control points. Clicking this option will switch to control line creation and change the mouse cursor icon. Clicking on any 2 control points will create a control line between them. For more information see the help for the '[ControlLine](#)' template.

## Tools Menu

The tools menu contains general operations for data management.

**GT Applications:** Launch a supported GT-SUITE application.



**GT-SpaceClaim:** Starts the GT-SPACECLAIM application. More information can be found in the [GT-SPACECLAIM Manual](#) or [Tutorial](#).



**GT-ISE:** Starts the GT-ISE application.



**COOL3D:** Starts the COOL3D application.



**VTDESIGN:** Starts the VTDESIGN application.



**GT-POST:** Starts the GT-POST application.



**CONVERGE Lite:** Starts the CONVERGE Lite application.





**Compare Files:** It is often useful and/or necessary to compare similar .ghx files. The user is constantly changing model attributes during the course of creating and running models. The "Compare Files" tool displays all of the parts that have dissimilar attributes between the two .ghx files.



**Reference Objects:** List all reference objects used in the model, and where they are used.



**Parameters:** List all parameters used in the model, and where they are used.



**Delete Unused Objects:** Delete all unused objects that are present in the model.



**Delete Unused Objects and Templates:** Delete all unused objects and templates that are present in the model.



**Delete Unused Templates:** Delete all unused templates that are present in the model.



**Refresh Implicit Object Links:** Update the attribute values of all objects that are linked to a library.



**Break All Implicit Object Links:** Break the implicit link of all objects that are linked to a library.



**Import Excel Object Blueprints:** Imports Excel file(s) to quickly create objects. In order for the object to be created correctly, the Excel file(s) must follow the proper format of templates provided in the .gto. The easiest way to get the correct file format is to right-click on a template and select the option to Generate Template Excel Blueprint.



**GT Excel Spreadsheets:** Access spreadsheets unique to GT-SUITE that can assist with data entry for a model.

## Component Menu

The Component menu will only be active when one or more components in the 3D canvas are selected, and the operation will apply only to the selected components.



**Cap :** Creates a cap (flow blockage) on pipes, Tsplits, and Ysplits. For more information see the help for the '[GEMCap](#)' template.



**Sleeve... :** Creates a sleeve (concentric pipe) over an existing pipe. For more information see the help for the '[GEMSleeve](#)' and '[GEMSleeveBend](#)' templates.



**Orifice... :** Creates an internal orifice in pipes, an orifice on the surface of a shell, or an orifice on a baffle. For more information see the help for the '[GEMOrifice](#)', '[GEMOrificePipe](#)', and '[GEMOrificePipeBend](#)' templates.





**Perforate Section...** : Creates a perforated section on pipes, sleeves, shells, flowsplits, and baffles. For more information see the help for the '[GEMPerfRef](#)', '[GEMPerfRefBend](#)', and '[GEMPerfCS](#)' templates.



**Perforate All...** : Creates a perforated section that covers the entire area of a baffle, Tsplit, and Ysplit. For more information see the help for the '[GEMPerfAll](#)' template.



**Perforate Point...** : Creates a perforated section at a specific location (point) on a shell. For more information see the help for the '[GEMPerfPoint](#)' template.



**Baffle Leakage** : Optionally adds a leak path at the intersection of the baffle and the shell.



**Baffle...** : Creates a baffle inside a shell. For more information see the help for the '[GEMBaffle](#)' template.



**Perpendicular Baffle to Baffle:** Creates a baffle perpendicular to an existing baffle. To do this first select an existing baffle. Then click on this command and the perpendicular baffle dialog window will open. A perpendicular baffle will be created at the specified distance from the center of the existing baffle. For more information see the help for the '[GEMBaffle](#)' template.



**Parallel Baffle to Baffle:** Creates a baffle parallel to an existing baffle. To do this first select an existing baffle. Then click on this command and the parallel baffle dialog window will open. A parallel baffle will be created at the specified distance from the center of the existing baffle. For more information see the help for the '[GEMBaffle](#)' template.



**Wool...** : Adds absorbing material (wool) to shells, sleeves, flowsplits, and individual chambers of shells. For more information see the help for the '[GEMWoolAll](#)', '[GEMWoolShell](#)', and '[GEMWoolChamber](#)' templates.



**Tank 3D Port** : Create a flow port on the surface of a 3D Tank.



**Sensor...** : Creates a sensor to sense quantities from the flow system. For more information see the help for the '[SensorConn3D](#)' template.



**Actuator...** : Creates an actuator to actuate quantities of the flow system. For more information see the help for the '[ActuatorConn3D](#)' template.



**Edit Flowsplit Ports** : Opens a 3D editor window for viewing and modifying inputs that characterize flowsplit ports.





**Move** : Allows a component to be moved along or rotated about its local orthogonal axes (X, Y, and Z). The operation can be done interactively or by specifying a distance or angle. To begin, first select the component to be moved and then click this operation. A small axis will be displayed at the components centroid. Multiple components can be moved in a single operation by clicking on another component to select it. To complete the operation press Esc or the [Stop Current Operation](#) option.

- Translation - To translate the component, click on an axis and drag the mouse. This will move the component along that axis. To define a specific distance to be moved, input that distance in the dialog window that opens in the upper left and press enter.

- Rotation - To rotate the component, click on the angle line and drag the mouse. This will rotate the component about that axis. To define a specific angle to be rotated, input that angle in the dialog window that opens in the upper left and press enter.



**Translation...** : Moves a component a specified distance. For more information see the help for the [Translation - Translates a component](#) dialog.



**Move Point-to-Point**: Moves a selected component(s) to a new location based on the relative location of 2 selected points. To begin, the component(s) to be moved must be selected. Then click on this command and the move point-to-point dialog window will open. Next the first point, or reference point, must be selected or explicitly entered. This reference point can be any control point in the model (it does not have to be on any selected component). Finally, the second point, or ending point, must be selected. This point can also be any control point in the model. The relative location of the 2 selected points will be calculated, and the selected component(s) will be translated by the same amount.



**Assembly Rotation**: Rotates an entire assembly in a specified plane. For more information, see the help for the [Assembly Rotation](#) dialog.



**Component Rotation...** : Rotates an entire component or assembly relative to the component that it is connected to. For more information see the help for the [Component Rotation](#) dialog.



**Scale Component**: Scales the volume of a component.



**Flow Port Direction...**: Assigns a flow direction to a port of the selected component. This will open a port selection dialog window that will allow the selection of which port to assign a direction to and the direction. During [discretization](#) GEM will use this direction when creating links between parts. This can be used to assign a flow direction to any port in the model that is not at a boundary (Add Connection>Subassembly connection should be used at boundaries). This is common for intermediate pipes inside of muffler shells as GEM won't know if they are inlets or outlets since they don't directly connect to the inlet or outlet set of pipes.



**Locate Connection on Tree**: Highlights the connection between the two selected components on the model tree. This tool is useful for converting the connection to a throttle, valve, etc.





**Convert Connection...:** Converts an existing connection into a different type of connection. This will open the port connection dialog window that will allow the selection of the new type of connection to be converted to. This should be used to change the type of connection between 2 components. The main use of this should be to convert an existing default orifice created from a flow connection operation into a non-default orifice or the desired type of connection.





## **CHAPTER 3: GEM3D Dialogs**

The dialogs section contains information on operations in GEM3D that require user interaction but do not have a template. For each dialog entry, a description of the operation is given as well as details about each item that is available from the dialog window. These descriptions are the same text that can be found in the context help for each dialog window. The context help can be viewed while using GEM3D by clicking the help button in each specific dialog window.





## Assembly Rotation - Rotates an entire assembly of connected components

This operation is used to rotate a set of components.

- 1) To begin an assembly rotation, first select the component to rotate, and then click the Assembly Rotation operation.
- 2) Next, the plane in which to rotate the component must be selected. For this step, GEM3D will automatically display the connection datum planes of the selected component, along with the local and global XY, XZ, and YZ planes. User-defined datum planes may also be used. Once the datum plane is selected, the assembly rotation dialog window will open.
- 3) Specify the angle to rotate the selected component. The assembly rotation operation makes the current orientation of the component the 0 point. Therefore, the angle specified in the dialog window is relative to the 0 point, or the original location of the components.

---

### Assembly rotation from reference

---

<b>Angle</b>	Specifies the angle to rotate the component in the selected datum plane. This angle is relative to the original orientation of the component before the assembly rotation operation was started.
--------------	--

- 4) Click OK or Apply to finish the rotation. The rotation will be performed by rotating the selected component in the selected connection datum plane. Any other components(s) connected to the selected component will be rotated as well to maintain any connection relationships.



## **Boundary Manager - Manages the connections at the boundaries (inlets and outlets) of the model**

This operation is used to manage all the boundaries and their connections. It is a convenient place to check, modify, and add boundary connections all in the same place. All connection types are allowed at boundaries, although the external subassembly connection is probably the most common. In fact, the application will automatically add external subassembly connections to any boundary port that did not manually get a connection added.

It is highly recommended to work through this manager at least once for each model so that all boundary ports are accounted for and are correct for the desired modeling activity. For example, this is a good place to verify (and modify if necessary) the flow direction expected at each boundary (inlets and outlets).

### **Boundary Manager Dialog Window**

---

<b>ID</b>	Displays the ID for each boundary connection. Each boundary ID will be displayed in the graphical window for easy identification.
<b>Location</b>	Displays the name of the component where the boundary connection is.
<b>Connection Name</b>	Gives the name of the connection component to be placed at this boundary. Each connection is treated like a reference object in that it can be opened with a double-click to be edited. A new connection can be added by typing a new name and double-clicking on it to create a new connection component.
<b>Do not create connections at boundaries</b>	Toggle to not create any boundary connections and just leave all boundary ports open (i.e. no connection). This is useful if the parts will be copied into another model or the model will be expanded (added to) before it is used in a simulation.





## Case Setup - Organizes cases and defines parameters

Case Setup is used to specify values for each parameter. Parameters are variables that are assigned by the user to attributes as the model is being built. (A parameter is designated by enclosing a parameter name in [square brackets] where variable data would normally be entered.) Parameters are typically used for two purposes:

- An attribute that will be changed frequently should be made a parameter so that it can be entered conveniently in Case Setup. Case Setup provides a common window to enter such data as well as provide a means of discretizing several model iterations at one time by assigning several different values to a given parameter.
- If several parameters in different components should have the same value, the same parameter name can be used in all of the attributes. Therefore, if a change needs to be made to this common value later, the change will need to be made in only one place. One limitation of this feature is that the unit types are checked to make sure they are consistent. For example, a parameter that is used to define a diameter (unit of length) may not be used to define an area (unit of length squared).

All of the parameters in the model will be listed automatically in Case Setup and each one must be defined for the first case of the model. Starting with the second case (the second column), only parameters that have different values than the previous case need to be filled. A check box at the top of each case allows the user to choose the cases that are desired to be discretized when the [Export .gtm](#) operation is done.

The asterisk (\*) is a special character that is allowed within Case Setup to signify that the parameter value will be set by a different model file (.gtm). The asterisk (\*) is only allowed for parameters that are used for attributes that do NOT affect the model geometry. When the model file is discretized, each parameter that has a value of asterisk (\*) will get a value of asterisk (\*) in the resulting .gtm model file. For more information on how the asterisk is handled in Case Setup of GT-ISE model files (.gtm), see the [GT-ISE Case Setup help](#).

### Active Case

Specifies the case number to be used when drawing the model in the graphical window. Only a single geometry can be drawn in the graphical window at a given time, and this drop down specifies which case to use to get the geometry. To update the graphical window after changing the **Active Case**, the Apply or OK button must be pressed.

### Case Label

Specifies a shortened, case specific, version of the legend. This specified label here will essentially be directly copied to the Case Label field in the Case Setup in GT-ISE. For more information about the proper use of the Case Label, see the [GT-ISE Case Setup help](#).

## Using Functions and Operators

Case Setup in GEM3D has all the same capability to use formulas and operators as Case Setup in GT-ISE. These operations are listed below. For a full description of these operations and how they work, see the [GT-ISE Case Setup help](#).

cos, sin, tan( $\theta$ )

acos, asin, atan()

pi()

sqr, sqrt()

min, max(a,b)

floor, ceil()

exp(a)

ln()



**log10()****logn(a,y)****if(condition,a,b)****Operators:** + , - , / , \* , ^**Scientific Notation Examples:** 1.2E-4 , 0.03E4 , 0.56E+4

Any cell can reference a cell to the left of it simply by typing in the command "`=<n`", where `n` is a positive integer which represents the number of cells to the left in the same row. Additionally a cell can reference another parameter for the same case. The button called Show Formula allows the user to see the formula that has been entered or likewise, the value of the equation.

**Modifying Strings of Text:** Strings of text and numbers can be prepended or appended by inserting the name of a parameter in brackets. For example, if `[RPM] = 1500`, entering "`[RPM]press.txt`" will return the string "`1500press.txt`". This feature is especially useful when specifying external file names that utilize specific naming conventions.

**Fixed Digit Integers:** To specify a specific number of digits displayed before or after a decimal, the following convention may be used in Case Setup: `[parameter_name%a,b]`, where `parameter_name` is the name of a parameter that represents a real number, `a` is a non-negative integer specifying the minimum number of digits to the left of the decimal, and `b` is a non-negative integer specifying the exact number of decimals to the right.



## **Center of Gravity - Calculates the center of gravity of components**

This operation is used to calculate the center of gravity (center of mass) of components. The center of gravity is the average location of all the mass in a particular component. It is equivalent to the center of mass. It is also equivalent to the center of volume since the density is assumed to be constant for all components. To calculate the center of gravity of a component, first select the component, then click on the Center of Gravity operation (on the Dimension menu). The center of gravity of multiple components can be calculated by selecting all the components and then selecting the Center of Gravity operation.

---

### **Center of Gravity Calculation**

---

<b>Name</b>	Displays the name of the component for which the center of gravity calculated.
<b>X</b>	Absolute X coordinate in the <a href="#">global coordinate system</a> of the center of gravity of the component.
<b>Y</b>	Absolute Y coordinate in the <a href="#">global coordinate system</a> of the center of gravity of the component.
<b>Z</b>	Absolute Z coordinate in the <a href="#">global coordinate system</a> of the center of gravity of the component.
<b>Unit</b>	Specifies the unit in which to display the X, Y, and Z locations of the center of gravity.





## Component Rotation - Rotates a component or assembly relative to the connected component

This operation is used to rotate a component a specified angle amount relative to the component that it is connected to.

1) To begin a component rotation, first select the component to rotate, then click the Component Rotation operation.

2a) If the selected component is only connected at 1 location, then that datum plane will be used and the component rotation dialog window will be opened directly.

2b) If the selected component is connected at more than 1 location (connection datum plane) then GEM3D will request that the user select the connection datum plane in which to rotate the component before opening the dialog window.

3) Specify the angle to rotate the selected component. The component rotation operation makes the current orientation of the components the 0 point. Therefore, the angle specified in the dialog window is relative to the 0 point, or the original location of the components.

---

### Component rotation from reference

---

**Angle** Specifies the angle to rotate the component in the selected connection datum plane. This angle is relative to the original orientation of the component before the component rotation operation was started.

4) Click OK or Apply to finish the rotation. The rotation will be performed by rotating the selected component relative to the component that it is connected to at the selected connection datum plane. Any other components(s) connected to the selected component will be rotated as well to maintain any connection relationships.





## Convert Shape to Component

This operation is used to convert a solid shape ('GEMSolidShape') or mesh shape ('GEMMeshShape') into a GEM3D component. This must be done for all shapes that need to be discretized because they will not be discretized during an [Export .gtm](#) operation.

### Port Selection

---

#### 3D Window

The first step in the conversion process is to select the ports which are important to the geometry. This should be done in the 3D window by right-clicking on detected ports to include or ignore them. The 'Include All Ports' and 'Exclude All Ports' buttons will add or remove all of the ports from the current selection.

#### Convert to

Specifies the type of component that the shape will be converted into. Depending on the particular component chosen, the conversion options may be different. For information on these component specific options, see the following tables for the desired component. Available components include:

- **Pipe:** Specifies the shape that will be converted into a pipe. This option requires 2 or 3 ports. Typically pipes will have an inlet and outlet, thus requiring 2 ports. Alternatively, a 3<sup>rd</sup> port may be included that is a single hole of a perforate section.
- **Flowsplit:** Specifies the shape that will be converted into a flowsplit. This option requires at least 1 port.
- **Shell:** Specifies the shape that will be converted into a volume which can be divided into smaller volumes. This option does not require any ports.
- **Multiple Flow Splits:** Specifies the shape that will be converted into a volume which is only divided by manual discretization planes, there is no automatic discretization from the x, y, and z dimensions.
- **Tank:** Specifies the shape that will be converted to a single flow volume with two fluid types. The relative volume of the fluid can change, and the surface orientation can change as a result of a 3D acceleration.
- **Array of Pipes:** Specifies the shape that will be converted into an array of straight pipes ('GEMMultiplePipe'). This option requires exactly 2 ports.
- **Pipe-T-Pipe...(Gallery or Rail):** Specifies the shape that will be converted into a series of alternating straight pipes and Tee flowsplits ('GEMSPipe' and 'GEMTsplit'). This allows for fast conversion of shapes such as a main gallery in an oil circuit, or a fuel rail. This option must have at least 2 ports.
- **Thermal Mass:** Specifies the shape that will be converted into a thermal mass.
- **Finite Element (Thermal):** Specifies the shape that will be converted into a thermal finite element mesh.







- **Flexible Body (Finite Element):** Specifies the shape that will be converted into a mechanical finite element mesh.
- **Rigid Body (Inertia3D):** Specifies the shape that will be converted into a mechanical rigid body (Inertia3D) component.





---

## Convert Shape Wizard: Pipe Conversion

---

### Pipe

When converting to a pipe, the following options are available.

#### Geometry Definition

Specifies the geometry of the pipe.

- **Bend Type:**
  - **Straight** models a straight pipe ('[GEMSPipe](#)')
  - **Single Bend** models one bend pipe ('[GEMBPipe](#)')
  - **Multiple Bends** can model the pipe with multiple bend section ('[GEMPipeXYZPoints](#)')
- **Direction for Straight and Single Bend types:**
  - **Vector between port centers (recommended)** indicates the GEM3D component will be converted such that the direction of extrusion will be along a vector connecting the center of the 2 ports. This method should be used when neither of the pipe ports are normal (perpendicular) to the flow direction of the pipe.
  - **Normal to port 1** indicates the GEM3D component will be converted such that the direction of extrusion will be normal to the selected port 1.
- **Cross Section Variation for Multiple Bends:**
  - **Linear interpolation between ends** indicates that a simple linear-interpolation is used along the length of the pipe between the two fixed cross sections specified for port 1 and 2.
  - **Multiple sections between ends** indicate that the cross section will change along the length of the pipe. This represents an accurate representation of the geometry and is typically the recommended choice.

#### Port Size Definition

Specifies the port size to be used for the created pipe. Choices include:

- **Define from Port 2/1** can be used to ignore any selections made for the current port, and duplicate the port size and shape measured at the other end of the pipe. Only one port can have this option selected.
- **Measurement Location:**
  - **End** indicates that the measurement location will be at the end of the part.
  - **Adjacent Pipe** will use the size of the neighboring part, if one is available.
  - **Custom Plane** will allow you to position a plane at a point along the pipe, and measure the size and shape





using that plane.

- **Shape:**

- **Measured Effective Diameter** converts the cross section area to a round shape with the equivalent diameter. This measurement is taken at the Measurement Location specified above.
- **User Defined Diameter** allows you to override the measured diameter, and enter a custom value. This can be especially useful when converting pipes from a mesh shape where the diameter is not a nominal size. For example, a pipe may measure 49.98 mm when the true value should be 50 mm.
- **Measured Rectangle Cross Section** will create a rectangular shape at the Measurement Location specified above. The Height and Width of the rectangle will be automatically determined from the pipe geometry.
- **Measured General Cross Section** will create a general cross section at the Measurement Location specified above. The cross section shape will be automatically determined to approximate the pipe geometry.

### Features

Specifies any features that will be converted at the same time as the base pipe. Choices include:

- **Perforates** when checked this option will allow a perforate section to be converted and added to the pipe. The additional steps required will automatically be included in the wizard after the base pipe conversion is complete. This option will be checked by default if a 3<sup>rd</sup> port was included in the port selection step. This option will be available whenever more than 2 ports exist for a pipe conversion. See the [perforate feature conversion](#) for more information.

### Calculated Geometry (after conversion)

Shows the converted geometry. Attributes include:

- **Geometry** data can be checked/changed.
- **XYZPoints** can be checked (for Multiple Bends only).
- **Cross Sections** cross sections and distance to next cross section can be checked/changed.
- **Location** of the converted component can be checked/changed.

### Name and Simulation Attributes (after conversion)

The name of the object must and object attributes can be specified/changed.





## Convert Shape Wizard: Perforate Feature Conversion

This step of the wizard converts a perforate feature from the model and adds it to the converted pipe. If a 3<sup>rd</sup> port was selected in step 1, the perforate feature option was included by default and that same port will default to be selected in this step. If desired, any new perforate port may be selected by right-clicking on any unused port and selecting "Include Port". If only 2 ports were selected in step 1, then a perforate port must be selected by right-clicking on any unused port and selecting "Include Port".

## Perforate Section

When converting a perforate feature on a pipe, the following options are available.

### Shape

Specifies the size of the hole of the perforate section.

- **Measured Effective Diameter:**
  - The effective diameter of the selected perforate hole will be measured from the model. All holes of the perforate section will be assumed to have the same hole diameter. A separate perforate feature can be added to the pipe after conversion if another section exists with a different hole diameter.
- **User Defined Effective Diameter:**
  - Allows you to override the measured effective diameter a custom value.

### Holes

Specifies the number of holes of the perforate section.

- **Measured Number of Holes:**
  - The number of perforate holes having the same effective diameter and in the same section of the pipe will be measured from the model.
  - *When using this option it is important to be sure that the 3D model has all the holes that would be created in the physical model. It is not uncommon for a 3D model to only contain a few holes of the perforate section for graphical performance. In this situation, the user defined option can be used to correctly set the number of holes that will be in the physical model.*
- **User Defined Number of Holes:**
  - Allows the measured number of holes to be overridden and a custom number of holes defined.
- **User Defined Porosity:**
  - Allows the measured number of holes to be overridden and a custom porosity value defined. The porosity is defined as the fraction/percentage of open area of the holes to the total area covered by the perforate section.

### Location

Specifies the location along the pipe where the perforate section is.

- **Measured:**
  - The start and end of the perforate section will be





measured from the model. The algorithm assumes that holes in the perforate section are placed within 2x the effective diameter of an individual hole. If the spacing of the holes is larger than this, it is necessary to use the custom start and end option to manually define the location.

- *While rare, it should be noted when using this option to ensure that the 3D model has the full location covered that would be in the physical model. In this situation, the custom start and end option can be used to correctly set the location that will be in the physical model.*
- **Custom Start and End:**
  - The start and end of the perforate section may be defined manually in the graphical window. When this option is selected a datum plane will be placed at each end of the pipe in the graphical window. These planes may be dragged and placed along the pipe to define the start and end of the perforate section.

### Coverage

Specifies the area around the pipe covered by the perforate section.

- **Measured:**
  - The angle of coverage and start of coverage of the perforate section will be measured from the model. The algorithm assumes a radial spacing of 2x the effective diameter of an individual hole. If the spacing is more than that, it may be necessary to use the user defined option.
  - *While rare, it should be noted when using this option to ensure that the 3D model has the full area around the pipe covered that would be in the physical model. In this situation, the user defined option can be used to correctly set the coverage that will be in the physical model.*
- **User Defined:**
  - Allows the measured angle of coverage and start of coverage of the perforate section to be overridden and given a custom value.





---

## Convert Shape Wizard: Flowsplit Conversion 3 Ports

---

### Flowsplit

When converting to a flowsplit with 3 ports, the following options are available.

#### Geometry Definition

Specifies the geometry of the flowsplit:

- **General**, '[GEMFsplitGeneral](#)':
  - The actual (true) surface area of the shape is calculated and used for the Surface Area attribute.
  - The actual volume of the shape is calculated directly from the geometry.
  - The characteristic length is calculated as the perpendicular distance from the associated port to a physical wall or another port.
  - The expansion diameter ( $D_{exp}$ ) is calculated from the characteristic length ( $L_{Char}$ ) of the associated port and the total volume ( $V$ ), assuming a cylindrical shape, using the

$$D_{exp} = \sqrt{\frac{4 \cdot V}{\pi \cdot L_{Char}}}$$

following formula;

- **Y Split**, '[GEMYsplit](#)' uses the ports to determine the diameter and angles of a Y flowsplit:
  - **Port Size Definition**

**Define from Port 1/2/3** Specifies the diameter from port 2/3 that will be used for port 1 and vice versa.

**Normal to port 1** indicates the GEM3D component will be converted such that the direction of extrusion will be normal to the selected port 1.
  - **Measurement Location**

**End** indicates that the measurement location will be at the end of the part.

**Adjacent Pipe** indicates that the measured effective diameter will be taken from the adjacent part.

**Custom Plane** will allow you to position a plane at a point along the flowsplit port, and measure the diameter using that plane.
  - **Shape**

**Measured Effective Diameter** converts the cross section area to a round shape with the equivalent diameter. This measurement is taken at the Measurement Location specified above.





**User Defined Diameter** overwrites the measured effective diameter.

- **T-Split, 'GEMTsplit'** (port 1 is the main diameter, port 3 is the T diameter) uses the ports to determine the main branch diameter and direction, and the port 3 diameter and rotation perpendicular to the main branch.

- **Port Size Definition** for port 1 and 3
- **Measurement Location**

**End** indicates that the measurement location will be at the end of the part.

**Adjacent Pipe** indicates that the measured effective diameter will be taken from the adjacent part.

**Custom Plane** will allow you to position a plane at a point along the flowsplit port, and measure the diameter using that plane.

- **Shape**

**Measured Effective Diameter** converts the cross section area to a round shape with the equivalent diameter. This measurement is taken at the Measurement Location specified above.

**User Defined Diameter** overwrites the measured effective diameter.

### Calculated Geometry (after conversion)

Automatically calculated geometry data can be overwritten:

- **Boundary Data** for flowsplit General only
- **Geometry / Location** for Y Split and T-Split

### Name and Simulation Attributes (after conversion)

The name of the object must and object attributes can be specified/changed.





## Convert Shape Wizard: Flowsplit Conversion 4 Ports

### Flowsplit

When converting to a flowsplit with 4 ports, the following options are available.

#### Geometry Definition

Specifies the geometry of the flowsplit:

- **General**, 'GEMFsplitGeneral'
  - The actual (true) surface area of the shape is calculated and used for the Surface Area attribute.
  - The actual volume of the shape is calculated directly from the geometry.
  - The characteristic length is calculated as the perpendicular distance from the associated port to a physical wall or another port.
  - The expansion diameter (Dexp) is calculated from the characteristic length (LChar) of the associated port and the total volume (V), assuming a cylindrical shape, using the

$$D_{\text{exp}} = \sqrt{\frac{4 \cdot V}{\pi \cdot L_{\text{Char}}}}$$

following formula;

- **X Split**, 'GEMXsplit' creates a flowsplit which uses the ports to determine the diameter and crossing angle for an X shaped flowsplit

#### Calculated Geometry (after conversion)

Automatically calculated geometry data can be overwritten:

- **Boundary Data** for flowsplit General only
- **Geometry / Location** for X Split

#### Name and Simulation Attributes (after conversion)

The name of the object must and object attributes can be specified/changed.







---

## Convert Shape Wizard: Flowsplit Conversion

---

### Flowsplit

---

When converting to a flowsplit with any number of ports, the following options are available.

#### Geometry Definition

Specifies the geometry of the flowsplit:

- **General**, 'GEMFsplitGeneral' (any number of ports):
  - The actual (true) surface area of the shape is calculated and used for the Surface Area attribute.
  - The actual volume of the shape is calculated directly from the geometry.
  - The characteristic length is calculated as the perpendicular distance from the associated port to a physical wall or another port.
  - The expansion diameter (Dexp) is calculated from the characteristic length (LChar) of the associated port and the total volume (V), assuming a cylindrical shape, using the

$$D_{\text{exp}} = \sqrt{\frac{4 \cdot V}{\pi \cdot L_{\text{Char}}}}$$

following formula;

#### Calculated Geometry (after conversion)

Automatically calculated geometry data can be overwritten:

- **Boundary Data** for flowsplit General only

#### Name and Simulation Attributes (after conversion)

The name of the object must and object attributes can be specified/changed.





---

## Convert Shape Wizard: Miter Bend Conversion

---

### Miter Bend

---

When converting to a Miter Bend, the following options are available.

#### Geometry Definition

Specifies the geometry of the bend:

- **Diameter of Larger Drilling:** This is the diameter that will be used for the main branch of the Miter Bend
- **Drilling Length:** This is the length of the main branch of the Miter Bend
- **Angle of Miter Bend:** The branch angle of the Miter Bend

#### Name and Simulation Attributes (after conversion)

The name of the object must be specified, and object attributes can be specified/changed.





## Convert Shape Wizard: Shell Conversion

### Shell

When converting to a Shell all detected ports are automatically neglected. A closed volume will be the result of a geometry conversion to a Shell.

### Shell conversion step

#### Cross Section Type

- **Create cross sections** creates a shell component using cross sections ('GEMShell') that represents the shape
- **Use imported shape** will use the imported shape geometry directly ('GEMSolidShell' or 'GEMMeshShell') for the shell conversion.

#### Cross Section Locations

- **Automatic** will detect cross sections automatically.
- **Custom** allows the location of each cross section to be specified by the user. Clicking this option will display the appropriate number of sections (given below) on the shape in the graphical window. Each cross section can then be placed at a custom location along the **Major Axis** by clicking and dragging with the mouse. Cross sections can be removed by right-clicking on the sections and choosing remove plane. Cross sections can be added by right-clicking on the sections and choosing add plane.

*Note: When using this option along with the **Custom axis** option for the **Major Axis**, the **Custom axis** cannot be changed. This option must be disabled to change the **Custom Axis**.*

This option can be used to reduce the number of cross sections needed to describe the shape accurately. Typically, shell shapes are complicated such that if cross sections are evenly spaced, it would require many, many cross sections to accurately describe the shape. By using this custom option and placing the cross sections at strategic locations along the shape, it greatly reduces the number of cross section needed to describe the shape. This has the benefit of making the resulting shell easier to work with.

#### Specify Cross Sections (Create cross sections only)

- **Major Axis** specifies the coordinate direction that is aligned with the major axis (main direction of flow) of the shape. The coordinate direction choices include:

**def** indicates that the converter will assign the major axis to the direction (only the X, Y, or Z directions) of the longest shell dimension (length, width, or height).

**X-axis** indicates that the converter will assign the major axis to the global X-axis direction.

**Y-axis** indicates that the importer will assign the major axis to the global Y-axis direction.

**Z-axis** indicates that the importer will assign the major axis to





the global Z-axis direction.

**Custom** indicates that a custom direction will be specified. When this option is selected, an arrow and plane will be displayed in the graphical window. To choose the custom axis, simply orient the arrow by clicking and dragging on the tip or tail. The final arrow orientation will represent the major axis to be used (the green plane is simply a visual aid and will not be used for anything).

*Note: The **Custom axis** cannot be changed when the **Cross Section Definition** option **Custom** is used. To change the custom axis, the **Cross Section Definition** must be set to **Automatic**. Change the **Custom** axis accordingly and switch back to **Cross Section Definition** option **Custom**. The cross sections will align with the **Custom** axis.*

- **Vertices per section:** Specifies the number of radial points to be used on all cross sections to define their shapes. These points will be spaced in equal angular increments around the center of the cross-section (e.g. if 90 vertices are chosen, there will be one point every 4 degrees). The number chosen should be sufficiently large to accurately define the shape of any cross section of the shell. The number of vertices requested must be between 10 and 200. The default value is 30.
- **Number of sections:** Specifies the number of cross sections that will be taken between the shell ends. The number of cross sections requested must be between 4 and 200. The cross sections will be taken perpendicular to the **Major Axis** specified. The cross sections will be evenly spaced along the **Major Axis**, unless the **Cross Section Definition** option **Custom** is used. The number of cross sections chosen should be sufficiently large to accurately define the shape of the shell.

The cross sections at the ends of the converted shell component will be modeled by taking the cross section of the shape at 5% of the length from the end cross section and the next closest one. Therefore, if geometry very close to the end of the shape needs to be modeled, two cross sections need to be positioned very close to one another at the ends.

- **Volume:** Specifies the internal volume of the shape in the units given by the **Unit** drop-down below. This value will be used to scale all of the dimensions of the shell so that the overall volume is correct. This scaling has the main purpose of making the internal volume of the shell accurate despite any approximations in describing the shell shape. If the **Volume** is set to "def" or "0", no scaling of the dimensions from the shape file will occur. In this case the volume will be determined from the shape assuming that it was created in the units given by the **Unit** drop-down.
- **Unit:** Specifies the unit that the shape was created with (The unit the





parent STL file was created in). If a **Volume** of 0 is specified above, then this is the unit that the converter will assume the shape was created with. If a non-zero **Volume** is specified above, then this specifies the unit for that specified **Volume**. The unit choices include:

- m<sup>3</sup>
  - cm<sup>3</sup>
  - mm<sup>3</sup>
  - in<sup>3</sup>
  - ft<sup>3</sup>
  - Liter
  - US-gallon
- **Conversion tolerance:** The tolerance controls the numerical scheme used in the converter. The value must be between "0" and "1". The default value of 0.01 (1.0%) will be sufficient for most shape conversions. If the conversion routine fails, the above attributes should be varied to see if the shape can be successfully converted. If the conversion routine still fails, increasing the tolerance may solve the problem and allow a successful conversion. This should only be done as a last resort since a larger tolerance can cause other problems for calculations after the shape is converted.

## Features

During conversions to a shell component, some features of the shell may be detected and converted along with the parent component. Features are automatically identified based on the shapes in the model and their relative position inside the shell. To include a feature in the conversion, simply turn on (check) the box next to the desired feature. If a feature is turned off (not checked), then it will not be included in the conversion. Possible features include:

- **Baffles**

---

## Calculated Geometry step

### Calculated Geometry (after conversion)

Calculated geometry data can be checked/changed.

- **Geometry**
- **Cross Sections**
- **Location**

---

## Name and Simulation Attributes step

### Name and Simulation Attributes (after conversion)

The name of the object must and object attributes can be specified/changed.





## Baffle Conversion - Geometry Selection step

---

### Shell Baffle #

An entry is provided for each shape detected inside the shell. For each detected shape, multiple options are available as follow:

- **Use Baffle** Check this option is the associated shape is a baffle and should be converted as a baffle. If the associated shape is not a baffle, then turn off (uncheck) this option and the shape will not be converted.
- **Orifice** Check this option if the included baffle has 1 or more holes (orifices) on it and they will be included in the conversion.
- **Perforate** Check this option if the included baffle is entirely perforated (perforated over its entire area) and it will be included in the conversion.. *Note that perforated sections must still be added manually after the conversion.*

### Location

Defines the location of the baffle along/inside the shell.

- **Measured** will detect the location automatically based on the shapes location relative to the shell.
- **User Defined** allows the location of each baffle to be specified by the user. Clicking this option will display a semi-transparent green plane at the measured location of the baffle. This plane can be translated to any location and the baffle will be created at that location. Sometimes this option is required to get the exact location correct. This can occur when the baffle uses complex geometry where it meets up with the wall of the shell. This can cause the location algorithm to be off depending on the exact geometry.

### Thickness

Defines the wall thickness to be used for the baffle.

- **Measured** will calculate the thickness of the baffle automatically from the geometry of the shape. The measured wall thickness is shown in the adjacent cell.
- **User Defined** allows the thickness of the baffle to be specified by the user. Clicking this option will use the wall thickness specified in the adjacent cell. Sometimes this option is required to get the exact baffle thickness. This can occur if the baffle is not exactly planar, but has some geometry in the axial direction, potentially likely near the shell wall for manufacturing.

## Orifice on Baffle step

---

An extra step will be included for each baffle that had the "orifice" option checked. Each hole that should be created can be selected in the graphical window (right-click). Multiple holes can be selected. Once all the desired holes are selected, clicking next will display the calculated geometry and allow for any changes.

### Location

Defines the location on the baffle of the orifice (hole).

- **Orifice Location X** gives the local X coordinate of the orifice on the baffle.
- **Orifice Location Y** gives the local Y coordinate of the orifice on the





baffle.

### Shape

Defines the effective diameter of the orifice.

- **Measured** will calculate the diameter of the selected orifice. The measured value will be shown in the adjacent cell.
- **User Defined** allows the diameter of the orifice to be specified by the user. Clicking this option will use the diameter specified in the adjacent cell.

## Perforate on Baffle step

---

An extra step will be included for each baffle that had the "perforate" option checked. One of the individual holes of the perforate must be selected in the graphical window (right-click). Only one hole should be selected. It is not necessary to select all the holes to be included. Once the single hole is selected, clicking next will display the calculated geometry and allow for any changes.

### Shape

Defines the effective diameter of a single hole of the perforate section.

- **Measured Effective Diameter** will calculate the diameter of the selected hole. The measured value will be shown in the adjacent cell.
- **User Defined Diameter** allows the diameter of the hole to be specified by the user. Clicking this option will use the diameter specified in the adjacent cell.

### Holes

Defines the effective diameter of the orifice.

- **Measured Number of Holes** will calculate the number of holes present on the baffle that have the same diameter as the selected hole. The measured value will be shown in the adjacent cell. *Note that this calculation will only be accurate if the imported geometry has the true number of holes to be manufactured. It is quite common for CAD files to contain a reduced number of holes for simplicity.*
- **User Defined Number of Holes** allows the number of holes to be specified by the user. Clicking this option will use the wall thickness specified in the adjacent cell.
- **User Defined Porosity** allows the porosity of the perforate section to be specified by the user. Clicking this option will use the porosity specified in the adjacent cell.

## Baffle Conversion - Options step

---

### Baffle Conversion

This step shows the calculated geometry of all the includes baffles, orifices, and perforates included in the conversion. The geometry and be reviewed and changed if necessary.





---

## Convert Shape Wizard: System of Pipes and Flow Splits

---

### System of Pipes and Flow Splits

This option is useful when an imported shape should be converted into a system of connected pipes and flow splits (['GEMFlowSystem'](#)). The discretization of the shape into separate parts is determined entirely by manually positioned discretization planes (child datum planes). This option is useful for modeling any flow path consisting primarily of hoses or pipes (intake/exhaust systems, coolant hoses, HVAC refrigerant lines, air ducting, oil passages, etc.).

This option provides an alternative to the traditional approach of cutting the imported geometry and then making separate conversions into the individual pipes and flowsplits. In this approach, all parts are created with a single conversion. The discretization planes are stored, along with the original geometry. This allows for efficient re-use of the planes (on new imported geometry) or easy adjustments to the planes for re-conversion.

#### **Name and Simulation Attributes**

The object name should be specified, along with the non-geometric attributes.

The object name will be used as the “base name” for all pipes and flowsplits created when the shape is converted. For example entering “AirSupply” will generate parts AirSupply-1, AirSupply-2, etc.

The attribute values entered will be applied to all pipes and flow splits that are created when the shape is converted. Note that some attributes apply only to pipes or flowsplits, and these are clearly noted in the attribute name.

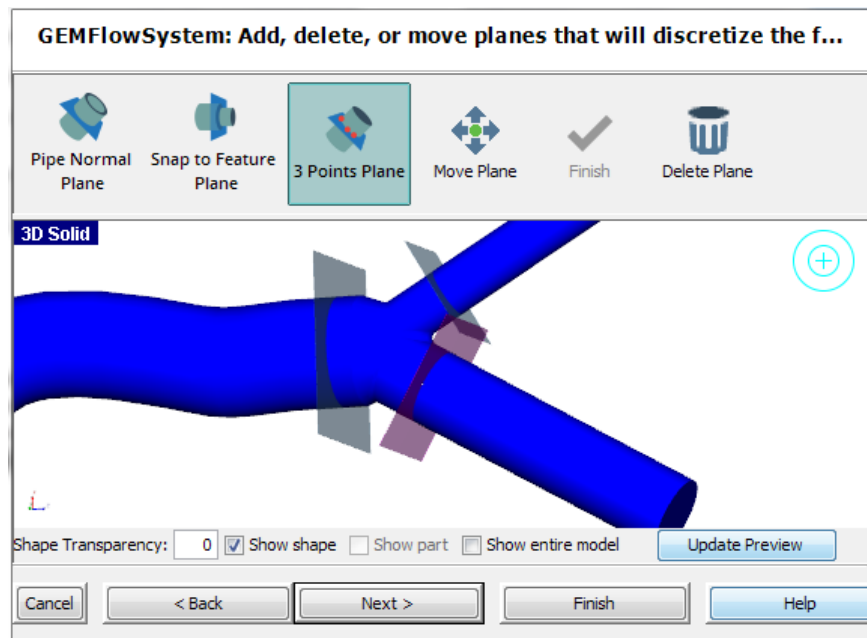
#### **Add, delete, or move planes**

In this dialog, discretization planes can be positioned on the imported shape at the locations where the shape should be divided into separate pipes or flowsplits. Any child datum planes that were positioned on the shape prior to starting the conversion wizard will be displayed and used as discretization planes. Planes may be moved or deleted as well.

The “Update Preview” button may optionally be used to preview the different parts that will be created at conversion.





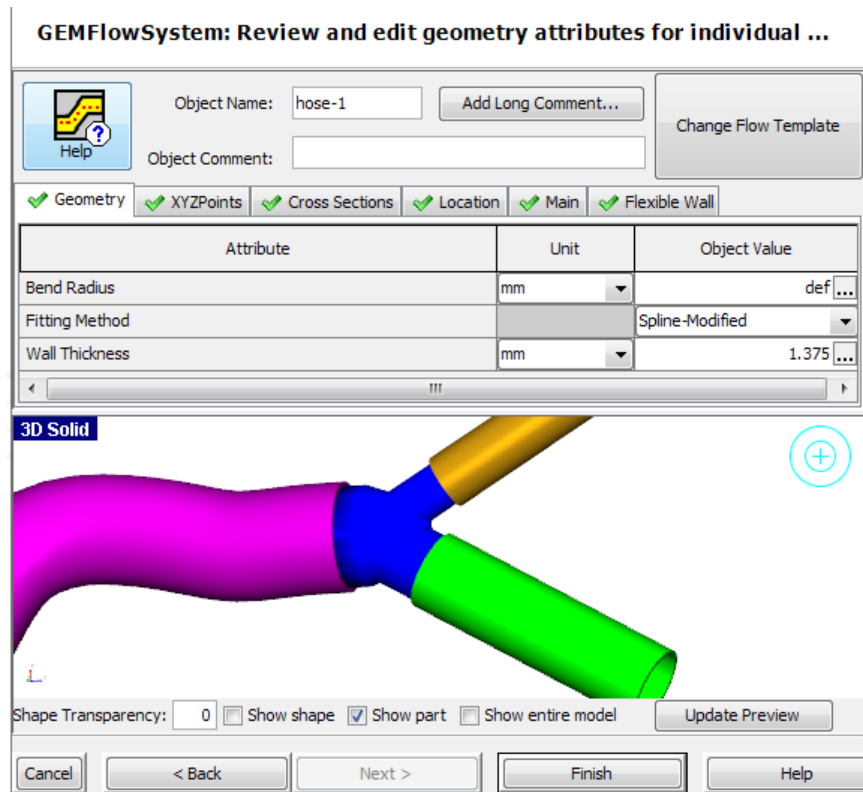


### Review and edit geometry attributes

In this dialog, the imported shape has been converted to separate pipes and flowsplits using the provided discretization planes. The geometric attributes for the highlighted part are shown in the folders at the top of the dialog, and may be edited if desired.

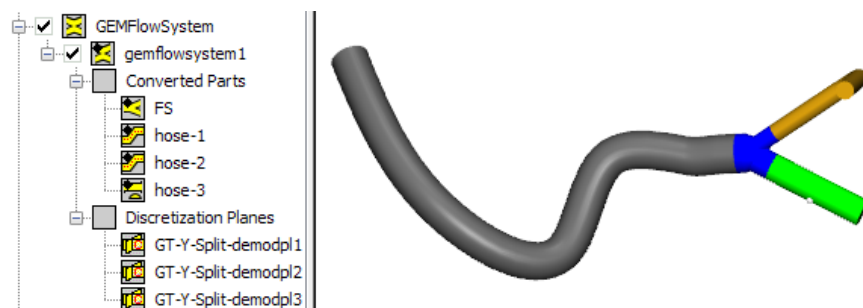
The part names are created using the base object name from an earlier dialog, but may be changed if desired.





During the conversion, an attempt is made to select the most appropriate template (straight pipe vs. bent pipe vs. flowsplit). However there may be some cases where it is desirable to change the template. This can be done using the “Change Flow Template” button.

At the completion of the conversion wizard, the '[GEMFlowSystem](#)' will be created in the model tree, containing the converted parts, original geometry, and the discretization planes.





---

## Convert Shape Wizard: System of Flow Splits

---

### System of Flow Splits

This option is useful when a solid shape should be converted into a system of connected flow splits (['GEMSolidFlowVolume'](#)). The discretization of the shape into flow splits is determined entirely by manually positioned discretization planes (child datum planes). This option is useful for representing engine cooling water jackets, or other geometries where there is a need to discretize the system into many flowsplits that don't conform to a nice grid such as that provided by the internal discretization available in the Shell option. Note that the Shell option also supports internal features such as baffles or wool, so it should be used for modeling mufflers or air boxes.

#### **Name and Simulation Attributes (after conversion)**

The object name should be specified, along with the non-geometric attributes.. These attributes will be applied to all flow splits that are created when the model is exported.

The geometry related attributes (not shown in the conversion wizard) depend on the placement of discretization planes (child datum planes), which may be added to the part after conversion. The values of these attributes (i.e. flow split volume) will be determined at model export.

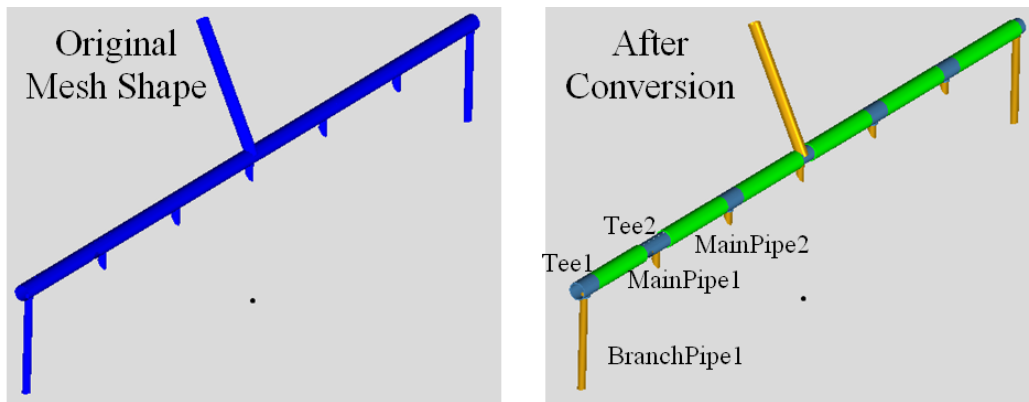




## Convert Shape Wizard: Pipe-T-Pipe Conversion

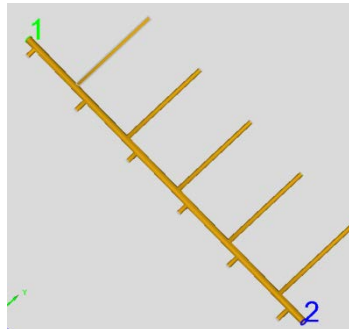
### Pipe-T-Pipe...(Gallery or Rail)

When converting a shape using the "Pipe-T-Pipe" option, the shape is converted into a series of pipes and flowsplits (typically alternating 'GEMSPipe' and 'GEMTSplit'). The figure below on the left shows an example of an original shape, and the figure on the right shows the series of pipes and flowsplits after the conversion. This allows for quick conversion of a shape comprised of several pipes and tees, as opposed to manually slicing and converting each individual pipe or flowsplit.



\*Important notes about conversion:

- 1) In order to convert properly, all ports (holes) must be open. This means that ALL branches of the shape must be open.
- 2) Upon conversion, Port 1 must be on one end of the main rail or gallery, and Port 2 must be on the opposite end, as shown in the figure below.



The options available in the dialog window when converting to a "Pipe-T-Pipe" are described below:

#### T-Pipe Conversion

- **Geometry Definition:**

- **Discretization Length:** The discretization length is used to set the cutting planes between each pipe and neighboring flowsplit. Typically for fuel rails and oil circuit galleries, this value should be around 10-20 mm. Each converted flowsplit will have a length equal to the discretization length defined here. This will then govern the length of each pipe along the rail.





- **Component Base Name:** Specifies the base name of each converted pipe and flowsplit. For example, if the base name was set to "Main", the pipe components would be named "MainP0", "MainP1", and so on, while the flowsplit components would be named "MainT0", "MainT1", and so forth.
- **Direction** Specifies the method that will be used to determine the starting point and direction of the GEM3D component. The choices include:
  - **Vector between port centers (recommended)** indicates the GEM3D component will be converted such that the direction of extrusion will be along a vector connecting the center of the 2 ports. This method should be used when neither of the pipe ports is normal (perpendicular) to the flow direction of the pipe.
  - **Normal to port 1** indicates the GEM3D component will be converted such that the direction of extrusion will be normal to the selected port 1. This is the default method and will typically provide the best conversion
- **Port Size Definition** Specifies the diameter to be used for the pipes and flowsplits along the main rail or gallery (i.e. Tee1 and MainPipe1 in the figure above). Choices include:
  - **Define from Port 2/1** Specifies the diameter from port 2 that will be used for port 1 and vice versa.
- **Measurement Location:**
  - **End** indicates that the measurement location will be at the end of the part.
  - **Adjacent Pipe** indicates that the measured effective diameter will be taken from the adjacent part.
  - **Custom Plane** will allow you to position a plane at a point along the pipe, and measure the size and shape using that plane.
- **Shape:**
  - **Measured Effective Diameter** converts the cross section area to a round shape with the equivalent diameter. This measurement is taken at the Measurement Location specified above.
  - **User Defined Diameter** allows you to override the measured diameter, and enter a custom value. This can be especially useful when converting pipes from a mesh shape where the diameter is not a nominal size. For example, a pipe may measure 49.98 mm when the true value should be 50 mm.

### Calculated Geometry (after conversion)



Shows the converted geometry. The choices include:

- **Geometry** where the geometry of the created parts can be changed.



**Name and Simulation  
Attributes (after  
conversion)**

- **Cross Sections** cross sections / diameter and distance to next cross section can be checked/changed.
- **Location** of the converted component can be checked/changed.

The name of the objects and object attributes can be specified/changed.





---

## Convert Shape Wizard: Multiple Pipes (Parallel)

---

### Multiple Pipes (Parallel)

Creates an array of multiple parallel straight pipes (['GEMMultiplePipe'](#)).

#### Geometry Definition

Specifies the geometry of the pipes.

- **Direction:**
  - **Vector between port centers (recommended)** indicates the GEM3D component will be converted such that the direction of extrusion will be along a vector connecting the center of the 2 ports. This method should be used when neither of the pipe ports is normal (perpendicular) to the flow direction of the pipe.
  - **Normal to port 1** indicates the GEM3D component will be converted such that the direction of extrusion will be normal to the selected port 1.

#### Port Size Definition

Specifies the port size to be used for the created pipes. Choices include:

- **Define from Port 2/1** Specifies the diameter from port 2 that will be used for port 1 and vice versa.
- **Measurement Location:**
  - **End** indicates that the measurement location will be at the end of the part.
  - **Adjacent Pipe** indicates that the measured effective diameter will be taken from the adjacent part.
  - **Custom Plane** will allow you to position a plane at a point along the pipe, and measure the size and shape using that plane.
- **Shape:**
  - **Measured Effective Diameter** converts the cross section area to a round shape with the equivalent diameter. This measurement is taken at the Measurement Location specified above.
  - **User Defined Diameter** allows you to override the measured diameter, and enter a custom value. This can be especially useful when converting pipes from a mesh shape where the diameter is not a nominal size. For example, a pipe may measure 49.98 mm when the true value should be 50 mm.
  - **Measured Rectangle Cross Section** will create a rectangular shape at the Measurement Location specified above. The Height and Width of the rectangle will be automatically determined from the pipe geometry.
  - **Measured General Cross Section** will create a general cross section at the Measurement Location specified





above. The cross section shape will be automatically determined to approximate the pipe geometry.

**Calculated Geometry  
(after conversion)**

Shows the converted geometry. The choices include:

- **Geometry** where the **Number of pipes** can be changed.
- **Cross Sections** cross sections and distance to next cross section can be checked/changed.
- **Location** of the converted component can be checked/changed.

**Name and Simulation  
Attributes (after  
conversion)**

The name of the object must and object attributes can be specified/changed.







---

## Convert Shape Wizard: 3D Tank

---

### 3D Tank

When converting to the Tank option ([GEMTank3D](#)), the solid geometry is used directly in the converted object. The component name is needed, and the simulation attributes (such as the fluid and thermal properties) can be changed during the conversion.

#### **Name and Simulation Attributes (after conversion)**

- The name of the object must be specified, and object attributes can be specified/changed.





---

## Convert Shape Wizard: Thermal Mass Conversion

---

### Thermal Mass

The Thermal Mass option can be used to convert solid shapes to a '[GEMThermalMass](#)'. There are no options for this geometry; the solid shape volume will be calculated and used in the converted template.

#### **Name and Simulation Attributes (after conversion)**

The thermal mass part name is required, the material properties and initial temperature can also be specified. These values will be included in the .gtm file when the model is exported. The display attributes in GEM can also be defined in this step.





---

## Convert Shape Wizard: Thermal Finite Element Conversion

---

### Thermal Finite Element

When converting to the Thermal Finite Element option ('GEMThermalFE'), the solid geometry is used as the basis for the finite element mesher. The surface mesh is shown in the conversion window, the completed volume mesh can be viewed after the conversion is complete or by opening the FE Mesh Reference Object in the Name and Simulation attributes screen of the wizard. The component name is needed, and the simulation attributes (such as the thermal properties) can be changed during the conversion.

#### Finite Element Mesh Inputs

- **Minimum Element Size:** This is the minimum element size that the mesher will typically use. Smaller elements may be used to preserve small geometry if it cannot be simplified.
- **Maximum Element Size:** The maximum size of the elements created. Increasing this number will create a coarser mesh, which will increase solver speed at the expense of temperature detail.
- **FE Mesh Object Name:** The name that will be used for both the mesh reference object and [GEMThermalFE](#) component.
- The name of the object must be specified, and object attributes can be specified/changed.

#### Name and Simulation Attributes (after conversion)





---

## Convert Shape Wizard: Mechanical Finite Element Conversion

---

### Mechanical Finite Element

When converting to the Mechanical Finite Element option ([GEMMechSolid3D](#)), the solid geometry is used as the basis for the finite element mesher. The surface mesh is shown in the conversion window, the completed volume mesh can be viewed after the conversion is complete or by opening the FE Mesh Reference Object in the Name and Simulation attributes screen of the wizard. The component name is needed, and the simulation attributes (such as the Mechanical properties) can be changed during the conversion.

#### Finite Element Mesh Inputs

- **Minimum Element Size:** This is the minimum element size that the mesher will typically use. Smaller elements may be used to preserve small geometry if it cannot be simplified.
- **Maximum Element Size:** The maximum size of the elements created. Increasing this number will create a coarser mesh, which will increase solver speed at the expense of model detail.
- **Limit on Error in Nodal Positions:** This attribute allows the mesher to more accurately follow curved surfaces in the geometry. By default, the value is 0, which lets the mesher automatically adjust the nodal positions. Note that 0 does NOT mean the error will be zero. But a value other than 0 will specify the maximum allowed error when the nodes are positioned. Values between 0.0001 and 0.01 mm are recommended, if a value other than 0 is necessary. This option is recommended to be used when modeling bearing surfaces.
- **FE Mesh Object Name:** The name that will be used for both the mesh reference object and [GEMThermalFE](#) component.
- The name of the object must be specified, and object attributes can be specified/changed.

#### Name and Simulation Attributes (after conversion)





---

## Convert Shape Wizard: Mechanical Rigid Body Conversion

---

### Mechanical Rigid Body (Inertia3D)

When using the Mechanical Rigid Body option ([GEMInertia3D](#)), a solid 3D CAD geometry is imported to carry out the conversion. The converted mechanical rigid body can be exported to GT-ISE as an Inertia3D object. This component automatically calculates the mass and inertia properties of the component. Additional nodes can also be defined on the rigid body which may serve as connection points for other components.

#### Material

- **Material Object:** The name of '[MaterialMechanical](#)' object describing the material properties of the component. This object will be used to calculate the mass and inertia properties. Standard materials are available in the template library, and they can be accessed by right-clicking and selecting "Value Selector".

#### Reference Node

- **Reference Node Label:** A label describing the reference node (port 0). This label will be used in the names of plots and data sets in GT-POST and will also be displayed when linking to the 'Inertia3D'.
- **LocalOrigin defining the Reference Node:** Name of the '[LocalOrigin](#)' object describing the reference frame in which the component's initial states are defined. If this attribute is set to "ign", the global (world) coordinate system of the CAD model is used as the reference frame. Note that the coordinate system described in the '[LocalOrigin](#)' object can be imported along with the original CAD model or created in GEM3D.
- **Initial X Velocity:** Initial X-component of the translational velocity of Reference Node with respect to the CAD Global Origin. ("def" = 0.0)
- **Initial Y Velocity:** Initial Y-component of the translational velocity of Reference Node with respect to the CAD Global Origin. ("def" = 0.0)
- **Initial Z Velocity:** Initial Z-component of the translational velocity of Reference Node with respect to the CAD Global Origin. ("def" = 0.0)
- **Initial Angular Velocity (X):** Initial X-component of the angular velocity of the body with respect to the CAD Global Origin. ("def" = 0.0)
- **Initial Angular Velocity (Y):** Initial Y-component of the angular velocity of the body with respect to the CAD Global Origin. ("def" = 0.0)





- **Initial Angular Velocity (Z):** Initial Z-component of the angular velocity of the body with respect to the CAD Global Origin. ("def" = 0.0)
- **Contact Geometry Object:** Name of the '[ContactGeom3D\\*](#)' object describing the surface associated with the rigid body at its Reference Node (port 0) for contact analysis purposes. If this attribute is set to "ign", no surface will be specified.

### Additional Nodes

- **Node Label:** A label describing each of the nodes. This label will be used in the names of plots and data sets in GT-POST and will also be displayed when linking to the 'Inertia3D' component.
- **LocalOrigin defining the Additional Node:** Name of the '[LocalOrigin](#)' object describing the local reference frame for the additional node. Note that the coordinate system described in the '[LocalOrigin](#)' object can be imported along with the original CAD model or created in GEM3D.
- **Export Nodal Position with respect to:** Two options are available for exporting the nodal position information of the local origin selected to define the position of the Additional Node:
  - **Reference Node** When this option is selected, the coordinates of the additional Node are defined with respect to the local reference node as defined in the Reference Node tab.
  - **Global Origin** When this option is selected, the coordinates of the additional Node are defined with respect to the CAD global (world) coordinate system.
- **Contact Geometry Object:** Name of the '[ContactGeom3D\\*](#)' object describing the surface associated with the rigid body at its additional node (Node #) for contact analysis purposes. If this attribute is set to "ign", no surface will be specified.





## Cross Section Editor - 2D editor used to create and modify custom cross sections

This editor is used to create and modify custom cross section shapes. This is done by drawing points in a 2D graphical window that are connected by straight lines to represent the shape.

The graphical window in the editor displays the current cross section (will be empty for a new cross section). The local cross section axis is drawn at the origin (0,0) to aid in building the cross section. A ruler is displayed at the top and left side of the graphical window to show the size of the graphical window. The current unit for the cross section is shown in the upper left corner of the graphical window. A coordinate indicator is shown in the upper right corner to show the current mouse location relative to the origin. The cross section can be moved in the graphical window by using the slider bars or by holding the shift key while pressing and holding the left mouse button and moving the mouse.

**Defining an Anchor Point:** The anchor point of a custom cross section is the reference point that will exist on the same extrusion line when cross sections are used to build components. The anchor point is set to the origin (X=0,Y=0) for new cross sections. The Center Polygon operation (see below) can be used to set the anchor point of the cross section to be the centroid (center of volume) of the shape. The anchor point is used when the custom cross section is used to build components, as the anchor point will line up with the anchor points of other pre-defined and/or custom cross sections.

The commands used to create and modify the shape are available on the toolbar above the graphical window.



**Undo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row.



**Redo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row.



**Toggle Grid:** Toggles the display of the grid. This also toggles the snap to grid option for points. When the grid is on, points placed or moved in the editor will snap to the nearest grid location. When the grid is off, then no snapping of points will occur.



**Move Group:** Allows the entire cross section to be moved.



**Add Point/Line:** Allows new points to be added to the cross section. When checked, clicking in the editor graphical window will place a point. An auto complete feature exists so that when future points are placed, a straight line will connect them to the last created point. To close the cross section, simply click on an existing point and a line will be created from the last point to the selected point, thus closing the shape. When not checked, no points can be placed making it easier to select and move parts of the cross section.





**Add Arc:** Allows an arc be added to the cross section. When checked, an arc can be added to the cross section by clicking 3 points. The first point will be used as the center of the arc. The second point selected will be used as the starting point for the arc. If this is not an existing point, then the auto complete feature will connect the last created point with this point using a straight line. The third point selected will be used as the end of the arc. Pressing and holding the control (ctrl) key will cause the arc to point in the opposite direction.



**Delete:** Deletes the current selection. This will delete the selected line or point from the cross section. Deleting a line simply removes the line. Deleting a point will also remove the 2 lines connected to this point.



**Clear All:** Clears the graphical window. This is equivalent to deleting all points and lines of the current cross section.



**Validate Drawing:** Validates the current cross section shape. This checks the shape to make sure it does not have any errors and is well formed. This option can also automatically close a shape by connecting the last placed point to the first point.



**Zoom In:** Zooms in on the cross section. [Keyboard hotkey = NumPad +]



**Zoom Out:** Zooms out from the cross section. [Keyboard hotkey = NumPad -]



**Zoom 1:1:** Restores the zoom to the default level (1:1).



**Fit to Screen:** Centers the cross section and zooms to the appropriate level to fit the entire cross section to the editor window.



**Center Canvas:** Centers the canvas on the origin without changing the zoom level.



**Edit Polygon Vertices:** Allows the points of the cross section to be built by giving the X' and Y' coordinates of each point. This can also be used to modify, add, or delete points from an existing cross section. For more information, see the help for [Polygon Vertices](#) command.



**Show Area:** Calculates the area of the current cross section.



**Rotate:** Rotates the cross section shape by a specified angle amount.



**Flip Horizontal:** Flips the cross section horizontally. The flip operation can only be done on closed cross sections.







**Flip Vertical:** Flips the cross section vertically. The flip operation can only be done on closed cross sections.



**Horizontal Symmetry:** Mirrors the entire cross section horizontally across the Y axis. This will essentially make a copy of the existing cross section and place it opposite the Y axis. This creates symmetry in the horizontal direction. The symmetry operation can only be done on open cross sections.



**Vertical Symmetry:** Mirrors the entire cross section vertically across the X axis. This will essentially make a copy of the existing cross section and place it opposite the X axis. This creates symmetry in the vertical direction. The symmetry operation can only be done on open cross sections.



**Center Polygon:** Centers the polygon by making the anchor point the center of volume and moving that anchor point to the origin.

**Unit:** Drop down menu specifying the units for the cross sections. Choices include m, cm, mm, in, ft, micron, km, mi, and dm.



**Configure Ruler:** Allows the ruler spacing to be manually specified.



**Set Anchor Point:** Allow the anchor point for the cross section to be manually specified. The anchor point is where all location, translation, and rotation operations will be measured.





## Cutting Plane Control Window - Controls the direction and orientation of the cutting plane

This control window allows the cutting plane to be manipulated in a specific direction or orientation. The cutting plane can still be controlled graphically using the mouse.

### Manage Plane Position

<b>Location</b>	Shows a real-time counter of the current X, Y, and Z location of the center of the cutting plane in the default unit (the ball of the arrow where it intersects the cutting plane).
<b>Orientation</b>	Shows a real-time counter of the current X, Y, and Z direction of the cutting plane as a unit vector (direction the head of the arrow is pointing).
<b>Translate</b>	Slider bar that will translate the cutting plane along the current orientation. Clicking outside of the control window will reset the position of the slider. The plane will be translated by a specified amount given by the setting of the <b>Translate plane</b> option below.
<b>Rotate X</b>	Slider bar that will rotate the cutting plane about the X axis (in the YZ plane). Clicking outside of the control window will reset the position of the slider. The plane will be rotated by a specified amount given by the setting of the <b>Rotate plane</b> option below.
<b>Rotate Y</b>	Slider bar that will rotate the cutting plane about the Y axis (in the XZ plane). Clicking outside of the control window will reset the position of the slider. The plane will be rotated by a specified amount given by the setting of the <b>Rotate plane</b> option below.
<b>Rotate Z</b>	Slider bar that will rotate the cutting plane about the Z axis (in the XY plane). Clicking outside of the control window will reset the position of the slider. The plane will be rotated by a specified amount given by the setting of the <b>Rotate plane</b> option below.

### Precision

<b>Translate plane</b>	Drop down option that determines how far the cutting plane translates with each move of the <b>Translate</b> slider above. The amount is given in units since the absolute amount will depend on the current default unit selection of the length.
<b>Rotate plane</b>	Drop down option that specifies how many degrees the cutting plane rotates with each move of the <b>Rotate</b> sliders above.

**Clip:** This button will do the Clip operation. This will separate the selected component(s) along the current location and orientation of the cutting plane.

**Reset:** This button will place the cutting plane back to its original position based on the 3 selected points that created it, effectively resetting the position of the cutting plane.





## Export Image - Export graphical view(s) to an image file

This command is used to export the graphical views to an image file.

<b>Export active view as image</b>	Exports the currently active view to an image file.
<b>Export all views in separate images</b>	Exports all views in the graphical display to separate image files, 1 for each view.
<b>Export all views in a single image</b>	Exports all views in the graphical display to a single image file. This will result in an image that looks exactly like the display window.
<b>White Background</b>	When checked, the exported images will use a white background color instead of the actual background color of the graphical window. Typically useful if the actual background is not white and the images are likely to be printed.
<b>Keep Ratio</b>	When checked, the ratio of the <b>Width</b> and <b>Height</b> will be kept constant as the values are changed. If the width is changed, the height will automatically change to keep the ratio constant (or vice versa).
<b>Width</b>	Width, in pixels, of the exported image.
<b>Height</b>	Height, in pixels, of the exported image.
<b>Output file</b>	Specifies the file name and location of the image file to be exported. Also allows a browse feature to choose the location and file name of the exported image file. By default the GEM3D model name will be used as the output file name. If no path is given, then the local working directory of the model will be used.

### Exporting an image file from the command line

In some situations, it may be desirable to export image files of a model from a command line (i.e. a DOS window on a PC and the command prompt in UNIX/Linux). The most likely situation for this is optimizations where the model is changed and discretized from the command line via a script or other interface. (To do this, use the command below. For a PC, the case (capitalization) is not important, but on UNIX/Linux it is:

General format: **gemc -im filename**  
 Example: **gemc -im model.gem**

This command will create an image file (jpeg) of the model and give it the same name as the base GEM file with a .jpg extension.

The help for the command line options can be found using the command **gemc -h**.





## Export Model - Export model file for use in GT-ISE

This operation is used to discretize the GEM3D model and export a model file (.gtsub or .gtm) for use with GT-SUITE.

When exporting the discretized model file, GEM3D will place the parts on the 2D map based on the current 3D orientation of the model in the GEM3D graphical window at the time of discretization (see the **Parts placement method** option below). The purpose behind this is to attempt to make the 2D map in GT-ISE look like the 3D model in GEM3D such that it is very easy to identify individual components. Of course, this task is quite difficult due to the 3D nature of the model and representing it on a 2D map. Whenever the 3D view would overlap components in a 2D map, GEM3D does its best to offset the parts on the map so the overall shape is maintained. The more complicated the 3D model, the more important the orientation is to maintain a clean looking 2D representation.

### Discretization

#### Pipe Discretization Length

Discretization length to be used for pipes during the discretization process. Even if a parameter is specified for the **Discretization length parameter** attribute below, this attribute must still be given a value. This is because there are specific discretization procedures that GEM3D does that require a valid numerical discretization length to be used. For example, when a pipe is perforated, this value determines how many flowsplits are used to represent the perforated length section.

#### Shell discretization length along X direction (local)

Specifies the target discretization length for shells in the local X direction (X direction of the shell's [local coordinate system](#)). In the X direction, the discretization length ( $\Delta X$ ) is constant. This length is calculated based on the largest shell dimension in local X direction and the user entered target discretization length, and again may be affected by certain orifice dimensions within the muffler.

#### Shell discretization length along Y direction (local)

Specifies the target discretization length for shells in the local Y direction (Y direction of the shell's [local coordinate system](#)). In the Y direction, the discretization length ( $\Delta Y$ ) is constant. This length is calculated based on the largest shell dimension in local Y direction and the user entered target discretization length, and again may be affected by certain orifice dimensions within the muffler.

#### Shell discretization length along Z direction (local)

Specifies the target discretization length for shells in the local Z direction (Z direction of the shell's [local coordinate system](#)). In this direction, each chamber of the shell is discretized separately, meaning that the discretization length ( $\Delta Z$ ) may not be constant along the entire shell. Within each chamber,  $\Delta Z$  is constant and is calculated based on the length of the chamber and the target discretization length, and may be affected by certain orifice dimensions within the shell. For instance, if there is a vertical pipe with an orifice diameter larger than the target Z discretization length,  $\Delta Z$  may be larger than if the pipe was not present.

#### Flowsplit acceptance ratio

Specifies the percentage (or fraction) of a cube's volume that must be contained inside the shell for that flowsplit to be retained. If a lower percentage of a cube's volume is contained in the shell, then GEM3D will





throw out (disregard) that flowsplit when discretizing the shell. If flowsplits are thrown out, the total volume of the shell or chambers will still be correct. In this case the total volume is divided evenly among the remaining flowsplits in the shell or chamber. The default value is 50 %.

This attribute should be used to retain normally thrown out flowsplits that would significantly change the flow path inside the shell. To illustrate, assume that a series of flowsplits are thrown out from a small section of the shell. If this small section of the shell represents a significant flow path that should be considered, then this attribute should be lowered so that those flowsplits are kept, thus retaining the significant flow path. It should also be noted that this attribute should not be changed to maintain accurate volume. The volume of chambers and shells will be correct even if flowsplits are thrown out of particular sections.

## Advanced Options

This button acts as a toggle to show/hide the advanced discretization options. The advanced options section contains options to enable/disable specific discretization rules. Generally these options are set to the best value recommend by GT and should not be changed. Particular models may require changing 1 or more of these options to provide the best discretization possible. Details on each of these options can be found in the **Advanced Options** table below.

## Output

### Model License Type

Specifies the license that will be used to run the exported GT-ISE model file (.gtm). Note that the license type chosen here can always be changed in GT-ISE if necessary.

### Output file name

Specifies the file name and location of the model file to be exported. Also allows a browse feature to choose the location and file name of the exported file. By default the GEM3D model name will be used as the output file name. An extension of .gtsub or .gtm is allowed.

- **gtsub** - The .gtsub extension is used to denote an external subassembly file. This extension must be used when the model contains external subassembly connections ('[SubAssExternalConn](#)'). This extension may be used even if the model does not have any external subassembly connections.
- **gtm** - The .gtm extension is used to denote a main model file. This extension may be used if the model does not have any external subassembly connections ('[SubAssExternalConn](#)').

### Append to file name (multiple cases)

Select an option below to adjust the filename of the discretized model when working with multiple cases.

- **Use Case Number** – The case number will be appended to the **Output file name** with the format of <filename>\_#.gtm.
- **Use Case Legend** – The case label (found in Case Setup) will be appended to the **Output file name** with the format of





<filename>\_CaseLabel.gtm.

### Part Spacing Factor

Specifies the relative spacing between parts on the GT-ISE map after exporting. The default value (“def”) is 1. This option must be greater than 0. Values less than 1 result in more compact models on the GT-ISE map, and values greater than 1 cause parts to be more spread out.

### Part placement method

The part placement method allows the user to specify the orientation of the GT-ISE model that is being exported. Two options are available from the drop-down menu:

- **Graphical orientation** will place the parts on the GT-ISE project map based on actual 3D shape. This is typically better for pipe systems with simple shells as the look of the model can be chosen and maintained by the user. Further information is included in the introduction section above this table.
- **Fixed** will place the parts in a fixed and regular manner on the project map. This is typically better when complex shells are present where there is no clear 3D orientation that will result in a good graphical placement.

### Use separate subassembly for each shell

The subassembly method determines how individual shells are created in the model file (.gtm) during discretization. Available subassembly methods include:

- **No** stores all discretized parts in a single model file for each case. This option is recommended for simple models containing a single shell or multiple simple shells. This was the default option that previous versions used.
- **External** stores each shell and any interior components and features into a separate external subassembly. This will create a separate model file (.gtm) for each shell present in the model and store it in the working directory. The main model will point to these subassemblies using a relative path. This is recommended for complete system models with a detailed shell or multiple shells when external subassembly references are desired.\*
- **Internal** stores each shell and any interior components and features into a separate internal subassembly. This will create a separate tab in the main model for each shell present in the model. This is recommended for complete system models with a detailed shell or multiple shells when internal subassembly references are desired.\*

*\* When using either the External or Internal subassembly methods above, the Parts Placement Method chosen will only apply to the main model. All subassemblies will use the "fixed" method as this will create a cleaner map for the many flowsplits that are created.*

### Export in V7.4 format

When checked, the .gtm model file will be written in v7.4 format. This will allow the discretized model file to be opened using GT-ISE v7.4. It will contain all parts and features that are fully compatible with GT-SUITE v7.4. This option will allow GEM3D v7.5 (and all its new features and benefits) to be utilized even if the GT-SUITE model must be run using v7.4.





Since GEM3D v7.5 has new features designed to make it better, this option may not always be possible. If the GEM3D model contains any new parts or features that are not compatible with GT-SUITE v7.4, an error message will be given during discretization saying that a v7.4 .gtm model file cannot be written. In this case, if a v7.4 .gtm model file is still desired, it is possible to remove the offending parts or features from the GEM3D model, then create the v7.4 .gtm model file. Then, the removed parts or features from the GEM3D model can be manually accounted for in GT-ISE v7.4.

## Preview Options

*The following 3 options affect the preview of the discretization. The discretization preview will discretize any shells in the model and figure out how many flowsplits are needed. Then, it will display all the flowsplits needed in the graphical window according to this preview option. The discretization preview is only done in a 3D view of the model (if only 2D views are used, the preview button will be disabled (grayed out)).*

*\*\*\*During the discretization preview, the user is allowed to display the flowsplit properties by double-clicking on any flowsplit in the graphical window. The flowsplit properties will show up as the properties of the flowsplit part that will be created on the GT-ISE project map. This helps to check the flowsplits main attribute values. Some of the flowsplit attributes are editable in the preview mode, as they are in GT-ISE (volume, characteristic length, expansion diameter). If the user changes those values in the discretization preview mode, they will be exported to GT-ISE.*

### Preview options

Specifies the type of preview option used to display the discretization preview in the graphical window. Available preview options include:

- **Show cube** displays each flowsplit as a separate cube. The size of the cube is proportional to the discretization length of the shell. All the cubes in a particular chamber will be displayed as the same color.
- **Show cube / connection** displays each flowsplit as a separate cube the same as the above option. Connections between the flowsplits (cubes) are displayed as black lines.
- **Show cube with volume** displays each flowsplit as a separate cube with different brightness. A lighter colored cube represented a larger percentage of the cube's volume being inside the shell. A darker cube represents a smaller percentage.
- **Show cube with mesh line** displays both the cubes and the mesh lines.
- **Show mesh line** displays lines that represent the edges of each flowsplit.
- **Show mesh face** displays planes (faces) that represent the division between flowsplits.
- **Show mesh face and line** displays both the mesh faces and lines.

### Cube transparency level (%)

Specifies the transparency level used when drawing the preview cubes (when using the cube preview option). 0 indicates opaque (solid) and 90 indicates almost completely transparent.







## Show model

Toggle to display the model during a discretization preview. When checked, the model will be shown in the graphical window with the discretization preview. When not checked, only the discretization preview will be shown.

## Advanced Options

*The following options are advanced discretization items that generally do not need to be changed. They are for advanced analysis of specific situations and for maintaining results from the previous version.*

### Lump small volume cubes into adjacent cubes

When checked, all cubes with a small volume will be lumped into adjacent cubes. This will avoid retaining flowsplits with small volumes (which restrict the time step), thereby improving simulation run speed. This will only be done in the local XY plane of discretization, so as not to alter the flow path in such a way that may produce incorrect results. The exact criteria for this small volume is given by the equation below:

$$\text{Volume} < 0.75 \cdot (\text{Flowsplit acceptance ratio}) \cdot (dx \cdot dy \cdot dz)$$

where  $dx$ ,  $dy$ , and  $dz$  are the imposed discretization lengths in the X, Y, and Z direction, respectively.

This option should be checked when discretizing mufflers (or other shells) that have many internal components that occupy a large portion of the total area and/or volume. This may be necessary when the internal components causes flowsplits to be thrown out (disregarded) that are required to make connections in the chamber. If such a situation occurs, GEM3D will automatically reduce its volume criteria to keep these flowsplits and maintain the connections. This has the side effect of creating smaller volume flowsplits than originally requested based on the discretization values.

This option should generally NOT be checked for large volumes that do not have many internal components. Without the internal components, connections in the chamber are not a problem and lumping the flowsplits will alter the flow path which may correspond to a change in results.

### Always keep flowsplits next to baffles

When checked, all cubes that are next to baffles (including end baffles on shells) will be retained and made into flowsplits. This option can be used to retain cubes for flow paths near baffles that may have otherwise been thrown out by the flowsplit acceptance ratio. This can potentially provide a better discretization if cubes next to baffles are removed that should connect to entrance and exit pipes. However, this option is not recommended for general discretization of models, only as a tool for specific models that have problems.

### Minimum segment length (fraction of discretization length) for PipeXYZ discretization:

This option is used for the discretization of GEMPipeXYZPoints components which use the Spline-Modified fitting method. An algorithm is used to determine the straight and bent segments of the pipe. Segments which are smaller than the fraction of the discretization length specified in this option will be combined with a neighboring segment. The default value is 75%. This option can be used to help reduce the







number of pipes shorter than the discretization length from a model. Note that the maximum segment length is the distance between Cross Sections in the Pipe XYZ Points component, each segment can only interpolate between two cross sections.

**Use face for shell  
flowsplit area calculation**

When checked, the expansion diameter for shell flowsplits along the major axis direction (local Z axis) will use the face area at the edge of the flowsplit. This allows for expansion diameters across flowsplits to change based on the true shapes geometry. Generally this option is the best representation of the actual geometry and should be used in all cases.

When not checked, the area at the center of the flowsplit will be used. Generally removing this option does not provide as good a result, but may avoid drastic area changes across flowsplits.

**Use improved expansion  
diameter rule**

When checked, the improved expansion diameter for pipes entering/exiting shells will be used. This improved rule does a better job of determining the proper expansion diameter based on specified discretization lengths and physical boundaries due to improvements in the 3D modeling algorithm. This is an improvement in all cases and is recommended for all models. New models will have this option checked by default.

When not checked, the standard expansion diameter for pipes entering/exiting shells will be used. This method tends to over-predict the expansion diameter when physical features have an effect.. This method is only recommended if matching previous versions results is desired.

**Use shape based  
characteristic length  
calculation rule**

When checked, the characteristic length for shell flowsplits will be determined by extending a vector in the port-normal direction until the boundary of the flowsplit is reached or a physical wall of the shape is encountered. The length of this vector will be set as the characteristic length. This is an improvement in all cases and is recommended for all models. New models will have this option checked by default.

When not checked, the characteristic length for shell flowsplits is determined by the distance available in the cube that represented that particular space. This method tends to over- or under-predict the characteristic length near the physical walls of shells. This method is only recommended if matching previous versions results is desired.

**Use multiple flowsplits  
for inlet/outlet pipe-shell  
connections**

When checked, the entering/exiting pipe will be connected to multiple flowsplits of the shell. This will distribute the flow more evenly in the shell so all the flow from the pipe is not forced into a single flowsplit. This will also improve the response of particular resonances as the pipe location does not have to be moved to connect to the center of only a single flowsplit. This is generally an improvement and is recommended in most cases. New models will have this option checked by default.

When not checked, the entering/exiting pipe will always connect to a single flowsplit. Due to this restriction, the location of the entrance actually can move within a discretization length since the pipe will





connect to the closest flowsplit at its center. This can sometimes cause flow restrictions and inaccuracies in the response of particular resonances. This method is generally only recommended if trying to match previous versions results, but can be used as a tuning factor.

*Note: This flag will usually only take effect if the entering pipe is larger than the discretization length (since if it is smaller it will only connect to a single flowsplit anyway). A warning message is given in the discretization output window (and the .msg file) during discretization if this situation is encountered.*

### Use new multi-dimensional discretization rules

When checked, more generalized multi-directional discretization rules for finding the areas and expansion diameters of shell flowsplits will be used. Generally this provides a better discretization for 3D shapes and is recommended.

When not checked, the discretization rules used in the major flow direction (local Z axis) are slightly different than what are used in the off directions (local X and Y axes). Generally this is not recommended since the GEM3D tool can build very general 3D shapes so the same rules should be applied in all directions. However, this option can be used as a tuning factor for models that appear to have a preferred direction.

### Use improved calculation of physical diameters in shells

When checked, an improved rule for calculating the largest physical diameter of entering/exiting pipes inside of shells will be used. This rule uses more sampling directions to better determine the largest physical diameter that would limit the flow expansion. This is an improvement in all cases and is recommended for all models. It has the largest effect when an entering/exiting pipe is near the edge of a shell and at a very large angle difference from the normal direction of the shell. New models will have this option checked by default.

When not checked, the previous rule will be used. This used 4 sampling directions to determine the largest physical diameter. This method is only recommended if matching previous versions results is desired.

### Use baffle area for expansion diameter calculation

When checked, the baffle area is used to determine the area available for expansion when flow passes through holes or perforates. When general baffles are used that are not oriented along a major axis of the shell (X, Y, or Z), this provides a better approximation of the available area to expand to. This is generally an improvement in most cases and is recommended for most models. It has the largest effect when the baffle has a large deviation from a local major axis. New models will have this option checked by default.

When not checked, the previous rule will be used, which used the cross section area of the shell at the baffle location. This calculation always used an area that was perpendicular to a local axis of the shell. This potentially could cause very small expansion diameters when the baffle did not deviate very much from a major axis. However, for large deviations this better approximates the true expansion area since the flow is likely to be along a shell major axis (although this depends heavily on





the actual geometry). This method is generally only recommended if trying to match previous versions results, but can be used as a tuning factor.

*Note: This flag will only take effect for general baffles that are not orientated along a major axis of a shell. If a model only has vertical and horizontal baffles, then this flag will have no effect.*

### **Use Port Area for Flowsplit Connection Diameter**

When checked, an algorithm is enabled to check for connections to flowsplits that require non-default connections and to add them. This involves 2 situations covered below. This is generally an improvement in most cases and is recommended for most models. New models will have this option checked by default.

- For any connection between 2 general flowsplits, if the minimum of the **Actual Port Opening Diameter** attributes is less than the minimum of the **Expansion Diameter** attributes, then an orifice connection will be created with the diameter equal to the minimum **Actual Port Opening Diameter**. Else, a default connection will be used.
- For a connection between a general flowsplit and another component, if the **Actual Port Opening Diameter** attribute is less than the minimum of either the **Expansion Diameter** or adjacent (pipe) diameter, an orifice connection will be created with the diameter equal to the **Actual Port Opening Diameter**. Else, a default connection will be used.

When not checked, the algorithm will not be used and default orifice connections will be used. This method is generally only recommended if trying to match previous versions results, but can be used as a tuning factor.

*Note: This flag will only take effect for connections involving general flowsplit. This does not affect the connections of flow components with shells.*

### **Automatically create discretization planes in shells**

When checked, temporary datum planes used for discretization will be created at all pipe ends that are inside of a shell. These discretization planes are the same datum planes that can be added manually, however, they are done automatically at discretization time. They are also temporary, so when the discretization process is complete, no datum planes are permanently saved. This has the advantage over manually creating planes as it would always be up to date for the current state of the model (manually created planes need to be updated with the model).

This is generally an improvement in all cases and is recommended for all models (see note below). New models will have this option checked by default. By aligning the discretization planes with the ends of pipes inside the shell, it ensures that the shell flowsplits will always line up with entering or exiting flow. This provides more accurate characteristic lengths, leading to more consistent results as the discretization size is changed.





*Note: It is possible that this feature could create very small volume flowsplits should a pipe end protrude beyond a baffle or shell end by a very small length. For example, if a pipe protrudes 5 mm past a baffle, this feature would cause a number of 5 mm long flowsplits to be created. This could cause an adverse effect on simulation speed. If this situation is encountered, it is recommended not to use this automatic feature and manually create only the discretization planes that are desired.*

When not checked, no automatic datum planes will be used. This method is only recommended if trying to match previous versions results.

*Note: This flag will only take effect if there are open pipe ends inside of a shell. This should affect just about every muffler, however a concentric tube resonator would not be affected. Many air boxes and plenums would be unaffected as there is no protruding pipes (they are typically flush with the shell wall or baffle).*

### Delete unused objects and templates

When checked, all unused objects and templates in the GEM model will not be included in the discretized GT-SUITE model file (.gtm).

When not checked, all objects and templates will be included in the model file. This matches the way previous versions worked.

Although the discretization occurs automatically, it may be important for the user to understand how it is performed. This section describes in general terms the rules that are used to discretize muffler elements.

Once the discretization lengths are determined, the shell is divided into slices in the local Z direction according to the calculated  $\Delta Z$  values. Each slice is further subdivided into rectangular sections based on  $\Delta X$  and  $\Delta Y$ . An algorithm determines the number of rectangles in each row and column that will most closely approximate the shell geometry. This results in the shell being divided up into many boxes that approximate the shape of the shell. Once the number of rectangular sections is determined, the total volume in that slice (minus the volume of any internal elements) is divided equally among the sections. The actual surface area of the shell is calculated and distributed appropriately to each flowsplit that is created based on its position in the shell. For this surface area calculation only the outer wall of the shell is used as this is the only part that is exposed to the environment. The resulting sub-volumes are placed on the GT-SUITE project map as 'FlowSplitGeneral' parts in a manner that visually represents their location in 3D space. Flow elements within the shell will be created as described in the following paragraphs, and will be placed on the project map beneath the shell.

By default, all shells in the model will be discretized according to the specified discretization lengths specified in this operation. Since multiple shells can be created in the same model file, it may be necessary to discretize some shells with different lengths. This can be done by overriding the discretization lengths specified in this operation. To override these values for a particular shell, the discretization values must be specified on the **Discretization** tab for each shell component that should be discretized differently.

It is sometimes necessary to create additional discretization planes inside of a shell. For this purpose, it is recommended to use the **Automatically create discretization planes in shells** advanced discretization





option above. Depending on the situation, it may still be necessary to manually choose a discretization point along the flow direction (local Z direction) for particular shells. This would provide the same discontinuity for the discretization that baffles do, however, without creating a physical division and disturbing the chamber. To handle this, the discretization routine will use discretization planes as a reference point when discretizing and start calculating a new discretization length in the flow direction perpendicular to the discretization plane. The discretization plane can be made from any existing child datum plane of the parent shell by checking the **Use as Discretization Plane** attribute in the '[DatumPlane](#)' template.

If the discretization length in any direction perpendicular to an entering pipe flow direction is less than the diameter (equivalent diameter if not round) of the entering pipe, then a situation could exist that may create an artificial flow contraction from the entering pipe into a single flowsplit of the shell. If such a situation exists, a warning message will be issued in the message window upon discretization. This is not an error, and the model will discretize as specified by the discretization window. This warning message is simply to alert the user of a potential artificial flow contraction based solely on discretization and not on a physical geometry.

Any solid section of pipe will be discretized according to its cross section shape. All '[GEMRPipe](#)' components and any '[GEMSPipe](#)' components with a circle cross section will be made into a single '[PipeRound](#)' object with the bend angle set to 0. Any '[GEMSPipe](#)' components with a rectangle cross section will be made into a single '[PipeRectangle](#)' object with the bend angle set to 0. Any '[GEMSPipe](#)' components with a rounded rectangle, an ellipse, or a custom cross section will be made into a single '[PipeCrossSection](#)' object with the bend angle set to 0. All of these will be discretized according to the user-entered **Pipe discretization length**. A new volume will begin at each internal orifice or at the end of a perforated section. The location of the ends of a pipe element will not necessarily coincide with the discretization points of the shell. Therefore, there are some rules that must be applied when connecting from a pipe to the shell volume. For pipe orifices, the connection will be made to the shell sub-volume containing the center point of the orifice. This means that all flow will travel into a single shell sub-volume (this is why the orifice size affects the discretization length).

Perforated sections of a pipe will be divided into sub-volumes of equal length and made into '[FlowSplitGeneral](#)' objects. The length of these sub-volumes will not exceed 1.5 times the user entered **Pipe discretization length**. For each sub-volume, the center point is determined along the length (in the flow direction). All of the flow through the perforate holes will be directed into a perpendicular plane that contains the center point. The perforate holes for each section will be represented by a number of orifices spaced equally around the perimeter of the pipe. The number of orifice objects used to represent the perforate holes is 4. An equal number of perforate holes will be assigned to each orifice. Each orifice will be connected to the sub-volume in the perpendicular plane that contains the appropriate point on the perimeter of the pipe. Therefore, each perforated sub-volume may connect to as many as four shell sub-volumes. The exception to this rule is the case where the perforate holes lead into a sleeve. In this case, the perforate holes are automatically assigned to one orifice rather than being divided.

Bent pipes follow the same general rules as straight pipes. Any '[GEMBPipe](#)' components with a circle cross section will be made into a single '[PipeRound](#)' object. Any '[GEMBPipe](#)' components with a rectangle cross section will be made into a single '[PipeRectangle](#)' object. Any '[GEMBPipe](#)' components with a rounded rectangle, an ellipse, or a custom shape cross section will be made into a single '[PipeCrossSection](#)' object. All of these will be discretized according to the user-entered **Pipe discretization length**. Any perforated section(s) are divided into equal sub-volumes that will become '[FlowSplitGeneral](#)' objects.





Pipes formed using X, Y, and Z coordinates follow the same general rules used for both straight and bent pipes. Any 'GEMPipeXYZPoints' components will be made into a single 'PipeTable' object. The resulting 'PipeTable' object will contain a series of straight and bent sections that best represent the geometry of the component. When the "Line-Arcs" fitting method is used, the resulting sections will match the geometry exactly as this method intentionally describes straight and bent sections. When using either "Spline" method, a best fit must be done to produce straight and bent sections. When doing this we prioritize geometries in the following order to produce a physically accurate result; 1) Total length of the entire pipe, 2) Total cumulative bend angle, 3) Total length of all bend segments, 4) Segment bend angles, 5) Segment bend radius, 6) Segment lengths. If desired, it is possible to create the straight and bent sections as individual parts using the "Discretize PipeXYZPoints as individual parts" option in [File>Options>Discretization](#).

A sleeve that has been added to a pipe will be divided into sub-volumes based on the sub-volumes of the parent pipe. These sub-volumes are further divided, if necessary, according to the user entered **Pipe discretization length**. Each of the resulting sub-volumes will become 'FlowSplitGeneral' objects or 'FlowSplitAbsorbing' objects if the sleeve contains wool.

'GEMTsplit' components (T-shaped components) are not discretized further and are represented by a single 'FlowSplitTRight' object in the project map.

A baffle orifice is handled in the same manner as a pipe orifice. All flow will be directed into the shell sub-volume containing the center point of the orifice. When there is a perforated section on a baffle, the number of holes will be distributed evenly among all of the shell sub-volumes that contact the perforated area.

Any open ports in the model file that require a default connection will be connected to a subassembly connection once discretized. If the open port requires a non-default connection, then this specific connection part will be created in the .gtm model file and not connected to anything. This will require a connection to be made to this non-default orifice before it can be connected to a subassembly connection or used in a model. Examples of this situation would be a pipe orifice at the end of a pipe that is at the end of the entire system or a perforated section that is not inside a shell.

### **Discretizing a model from the command line**

In some situations, it may be desirable to run the discretization of a model from a command line (i.e. a DOS window on a PC and the command prompt in UNIX/Linux). To run the discretization from the command line a complete .gem file must be used. To start the discretization from the command line, type the command below. For a PC, the case (capitalization) is not important, but on UNIX/Linux it is:

General format:	<b>gemc [OPTIONS] filename</b>
Specific format:	<b>gemc -VM.m.s -o:filename.gtm -p:parametername=value filename.gem</b>
Example:	<b>gemc -V2016 -o:output.gtm -p:diameter=50 model.gem</b>

Available [OPTIONS]:

-VM.m.s	Specifies the software version where M.m.s is the software version desired in the format of MajorVersion.MinorVersion.SubMinorVersion, for example, 2016.
-o:filename	Specifies the filename to be used for the output file created by GEM3D. The







	output of this command will be a model file for use in GT-ISE, so a .gtm extension is required. This option is not required. If no filename is given, then the model file will receive the same name as the .gem file (just with a .gtm extension).
-ov:M.m.s	Specifies the software version that the output file (.gtm) will be written to. This is the equivalent of the <b>Export in V#. # format</b> option when running from the graphical interface. The version is specified where M.m.s is the software version desired in the format of MajorVersion.MinorVersion.SubMinorVersion, for example, 7.4.0.
-p:parametername=value	Specifies the parameter name and value to be used during the discretization of the model. This can include any parameter defined in case setup in the GEM3D model. The value must be a valid entry for the particular attribute. Since there is no GUI to validate entries, if an invalid value is used, the discretization will simply fail. Multiple parameters can be changed by separating the entries with a colon (:) (-p:param1=value1:param2=value2).
-c:filename	Specifies the filename to be used that has the different configurations that are to be created during discretization. The extension of this file should be text format (.txt). This file can include any attribute or object that already exists in the model. This option is useful when trying to create multiple discretized models from a single 3D model to test different components. The general format of the configuration file is below
-m	Specifies that a 3D model file (.gem) will be created for each configuration. The model file name will be taken from the input GEM3D file. This option must be used carefully as it may overwrite the existing .gem file that was used as an input.
-ac:#	Specifies the active case of the input model file to be used during discretization. If the input model file contains multiple cases, this option specifies which case to use for all parameters values that are not passed using the parameter name option above. If this option is not used, then the current active case specified in the input model file will be used.

The following options are used only by themselves, without other options or a filename. The version flag may be included.

-help or -h                      Shows script help.

### Using a configuration file

General format:                      **Partname.AttributeShortName = Variable [Unit]**

**Partname**                              This is the name given to the object in the model tree.

**AttributeShortName**                      This is the name of an attribute that is given by GTI and is used by the discretization routine. The short name of an attribute can be found by clicking on the attribute name once.

**Variable**                              The new variable that is to be used for this configuration.





**Unit** If applicable, the unit of the new variable. This only applies to attributes that have units assigned to them.

An example of the configuration file format is below, along with a description of each line in { }. The brackets are not needed when creating the configuration file.

```
Discretization.configuration = 1      {this is the start of a new case and is required}
Discretization = case1.gtm           {output filename with .gtm extension of discretized model}
Muffler.SHELLDISCLENNGTHX= 40 mm    {shell discretization of the X-axis}
Muffler.SHELLDISCLENNGTHY= 40 mm    {shell discretization of the Y-axis}
Muffler.SHELLDISCLENNGTHZ= 40 mm    {shell discretization of the Z-axis}
Muffler.FLOWACCRATIO= 50 %          {acceptance ratio of the shell discretization}
MufflerBaffle.LOC= 217 mm           {changing the location of a baffle in a shell}
...
BafflePerf.NHOLE= 90                 {changes the number of holes on a perforated baffle}

Discretization.configuration = 2      {this is the start of a new case and is required}
Discretization = case2.gtm           {output filename with .gtm extension of discretized model}
MufflerBaffle.LOC= 250 mm            {changing the location of a baffle in a shell}
...
BafflePerf.NHOLE= 180                {changes the number of holes on a perforated baffle}
```

It is also possible to comment out lines in the configuration file so that they are not used during the discretization. To comment out a configuration just add // before the line. The format is below.

General format:           **//Partname.AttributeShortName = Variable [Unit]**





## Export STL - Export model to an STL file

This operation is used to export the model as an STL file. This can be done with the entire model or selected components.

<b>Output file name/location</b>	Specifies the file name and location of the STL file to be exported. The <b>Output To</b> button allows selection of the location and file name using a file browser dialog window. When the <b>Output separate STL files</b> option is not checked, this field specifies the output file name and its location. When the <b>Output separate STL files</b> option is checked, this field only specifies the output files location.
<b>Selected components only</b>	When this option is checked, only the selected components in the model will be exported to the STL file. When not checked, the entire model will be exported to the STL file.
<b>Output separate STL files</b>	When this option is checked, each component in the model will be exported as a separate STL file. The STL file names will be taken from the components name in the model file. When not checked, all components will be exported to a single STL file.
<b>Resolution</b>	Slider bar giving the effective resolution of the output STL file. The value can be from 0 to 100, with larger numbers signifying better resolution (more detail). At greater resolutions, smaller triangles are used to create the STL file and thus giving greater accuracy. This also produces more triangles and thus creates larger size STL files.



## Export ACIS - Export model to an ACIS file

This operation is used to export the model as an ACIS file. This can be done with the entire model or selected components. Note that Mesh Components (such as the 'GEMFsplitGeneral', 'GEMMeshShape', or 'GEMMeshShell') cannot be exported to ACIS format.

<b>Output file name/location</b>	Specifies the file name and location of the ACIS file to be exported. The <b>Output To</b> button allows selection of the location and file name using a file browser dialog window. When the <b>Output separate files</b> option is not checked, this field specifies the output file name and its location. When the <b>Output separate files</b> option is checked, this field only specifies the output files location.
<b>Selected components only</b>	When this option is checked, only the selected components in the model will be exported to the ACIS file. When not checked, the entire model will be exported to the ACIS file.
<b>Output separate files</b>	When this option is checked, each component in the model will be exported as a separate ACIS file. The ACIS file names will be taken from the component's name in the model file. When not checked, all components will be exported to a single ACIS file.
<b>Version</b>	Specifies the ACIS version to use for the exported file. Newer versions will have full support for all geometry types. Newer versions are not backward compatible, so it is possible that older versions of CAD or viewer applications may not be able to open newer versions. Choices include: <ul style="list-style-type: none"> <li>• R25</li> <li>• R2016</li> </ul>



## Filter Shapes - Filter shapes

This operation is used to quickly filter the shapes (mesh and solid shapes) in the model and determine which ones are actual components and which ones need to be deleted. The purpose of this operation is to make it easier to delete shapes that are not useful in the modeling process. This includes flanges, brackets, tabs, bolts, and other features in the file that are not useful when trying to create a flow model of the system. This operation will pass all shapes through a filter to create a list of shapes that are not likely to be necessary flow components, and thus can be deleted. To fail the filter validation the shape must meet the following 3 criteria.

- 1) It must be small in length, width, and height (less than 1 mm).
- 2) It must have no volume.
- 3) It cannot connect to another shape that does not meet the above 2 criteria.

<b>Name</b>	Lists the name of each shape that did not pass the filter validation. Clicking on a shape name or names in this list will display the component as selected in the graphical window. This allows the shape to be identified graphically to ensure this component needs to be filtered.
<b>Tolerance</b>	This slider bar sets the filter tolerance. A full negative (-) tolerance is extremely strict. This means that it will follow the above criteria exactly, without exception. This tolerance will result in only a few shapes failing the filter validation. However, the shapes that did fail have a very low probability of being actual components, thus preventing accidental deleting of shapes. Any level greater than full negative (-) will allow a certain tolerance when checking the above criteria (first 2). A full positive (+) tolerance is extremely loose. This tolerance will result in nearly all shapes failing the filter validation, including those that are actual components. The update button needs to be pressed every time the tolerance is changed to update the list of shape names.



## Import 3D - Import 3D file

This operation is used to import 3D files into a new or existing model file. Supported formats include **ACIS** (.sab, .sat), **SpaceClaim** (.scdoc), **NX** (.prt), **STEP** (.stp, .step), **STL** (.stl), **IGES** (.iges, .igs), **Parasolid** (.x\_t, .x\_b), **CATIA** (.catpart), **JT** (.jt), **Pro/E** (.prt), and **NASTRAN** (.nas). ACIS, NASTRAN, and STL files are directly imported by GEM. The other formats are imported using GT-SpaceClaim as an intermediate step and are automatically converted into GEM format. For complete information on each format, see the table at the end of this section.

*\*Note: Since GT-SpaceClaim is not supported on Linux (SpaceClaim is a Windows only application), any format that requires GT-SpaceClaim as an intermediate step can't be opened on Linux. Therefore, the file browse window will not show any of these file types.*

## Import 3D File

### Files to Import

This field specifies the 3D file to be imported. Any file type may be selected as GEM will automatically use GT-SpaceClaim for the required formats. Multiple files can be specified, but they must all be of the same type.

Clicking Finish will import the selected 3D file(s) using all default options. To use custom options, click Next. The wizard will offer different options depending on the file format selected. ACIS files will open the [Import ACIS](#) dialog window. STL files will open the [Import STL](#) dialog window. Other formats that go through GT-SpaceClaim do not have any custom options.

### CAD Formats for Import to GT-SUITE, listed in order of preference

File Format	Instructions for Import
Preferred Format	
ACIS	This format can be imported directly to GEM3D and COOL3D. Files can also be opened in GT-SPACECLAIM for modification if necessary.
Native CAD Formats	
SolidWorks	Files must be imported to GT-SPACECLAIM. The file must then be saved to ACIS format and imported to GEM3D or COOL3D.
NX/Unigraphics	
CATIA	Files must be imported to GT-SPACECLAIM. These file types require a special GT-SPACECLAIM license, at a modest cost, before they can be imported. They must then be saved to ACIS format and imported to GEM3D or COOL3D. Please contact <a href="mailto:support@gtisoft.com">support@gtisoft.com</a> for details.
Inventor	
Pro/Engineer	
Finite Element Formats	
NASTRAN	Files can be imported directly to GEM3D. Once in GEM, NASTRAN surface meshes can be converted to STL format (Mesh Shape), which opens the possibility of conversion to many GT components. NASTRAN volume meshes can be converted directly to thermal or mechanical finite element parts.



Data Exchange Formats	
<b>STL</b>	Files can be imported directly to GEM3D and COOL3D. This format is more suitable for single components; for compound components the ACIS or Native CAD formats would be better. File sizes can also become large if a very fine resolution is used when creating the .stl.
<b>STEP</b>	Files must be imported to GT-SPACECLAIM. The file must then be saved to ACIS format and imported to GEM3D or COOL3D.
<b>IGES</b>	Files should be imported to GT-SPACECLAIM. The file must then be saved to ACIS format and imported into GEM3D or COOL3D. Due to the possibility for missing surfaces with this format, ACIS or Native CAD formats are recommended.



## Import STL - Import STL file

This operation is used to import an STL file. There are 2 different methods of importing an STL file into GEM3D.

---

### Import STL As

#### Surface

This method imports an entire STL file based on its surfaces. This method is extremely accurate at representing the exact shape and dimensions of an STL file. Any STL file imported using this method must first be converted into GEM3D components before it can be discretized. This option is the default recommended option when importing STL files and should always be used unless a specific circumstance calls for the cross section method below. Choosing this option will open the [Import STL as Surface](#) dialog window.

#### Cross Sections

This method imports a shell from an STL file. This method determines a major flow direction, then represents the STL file by fitting cross sections normal to this major direction. This method should only be used if the STL file contains only a shell, without any other parts or features. Choosing this option will open the [Import STL as Cross Sections](#) dialog window.



## Import STL as Surface - Import an entire STL file from its surfaces

This operation is used to import the entire contents of an STL file into GEM3D. During the import routine, all the surfaces of the STL file are determined and used to create the shape in GEM3D. Each section of the STL file (see the **Split by connection** option below) will be imported as a 'GEMMeshShape' component. If the model file is empty, then the graphical view will automatically be reset to show the imported shape. If the model file is not empty (contains already existing components), then existing graphical view will be left alone. This can result in the imported shape not being shown because its absolute location is off the screen (see the **Center at Origin** option below). The 'GEMMeshShape' components cannot directly be discretized into flow components for use in GT-SUITE, so after the import operation they must be divided and converted into GEM3D components for discretization. For details about dividing and converting the mesh shapes, see the operations on the Convert menu.

The ability to divide and convert imported shapes depends very strongly on the quality and nature of the geometry of the STL file. The following guidelines should be followed to ensure the dividing and converting process is robust and consistent. Even if the guidelines are not followed, the mesh shapes may still be able to be handled (divided and converted), but the chance of success goes down.

- All holes in surfaces should represent flow ports. Holes in surfaces that will not be used as a flow port could potentially cause errors during conversion. If unwanted holes do exist, it may be possible to use the Patching operation to patch (close) these unnecessary holes.
- Ideally, the STL file should not contain wall thickness. In other words, it should represent a single inside surface. If the shapes do contain wall thickness, the outer wall can usually be removed with the Separate by Curves operation or the Marking operation.
- The STL file should represent the inside surface (that the flow sees) of the final assembly (or single component). In other words, if multiple mesh shapes need to be combined together to create a component, the chance of success goes down. For example, a pipe component should be created from a single mesh shape, and not from 2 mesh shapes that may represent two halves of the pipe. If this situation exists, then the Patch Mesh Ports operation may be able to help, as well as other [Convert Menu](#) operations. A GEMMeshShell - Mesh Shell component also will usually work in these situations.

## Import Options

### Split by connection

When checked, the imported shape will contain multiple sections based on the connection of triangles in the STL file. This should be checked if the STL file contains multiple separate sections, and these separate sections should be created as different components in COOL3D. When not checked, the imported STL file will be a single component, regardless of what connections exist in the STL file.

### Center at Origin

When checked, the centroid of the imported component will be placed at the origin (0,0,0). This should be checked if the absolute location doesn't matter and it would be beneficial to have the entire model centered. When unchecked, the STL file will be placed exactly as it was when the file was created. This should not be checked if the absolute location of the model is important and should be retained. If the absolute location is retained, then the imported shape will be placed at that exact location. If



there are existing components in the model, then the current graphical view will be maintained, which could result in the imported shape being off the screen.

### STL Reduction Fraction

Specifies a fraction of the total triangles present in the true STL file that are used to draw/display the shape in COOL3D. A value of "ign" (default value) or 1 will use exactly the true STL file. Any positive integer between 0 and 1 may be used to specify the fraction of the total triangles to use. The smaller the fraction, the smaller the model size (faster the application) but the less accurate the representation of the STL is.

The purpose of this option is to reduce the amount of data in the model (file size) to increase the speed of the application and all operations (rotations, marks, cuts, conversions, etc.). The trade-off to gain speed is less accuracy due to some of the triangles being removed. Generally, this option should not be used (value of "ign") since accuracy is desired and most STL files will not cause the application to be noticeably slower. This option may be used to reduce the model size if the STL file is very large (large meaning lots of triangles, not necessarily file size) and it is possible this could reduce application speed. This option should be used if the application is very slow, unresponsive, locks up, or runs out of memory when importing a full STL file (value of "ign"). It is difficult to tell if this option is needed prior to working with the STL in COOL3D. The general recommendation is to first not use this option because it is usually very quick and easy to tell that the application is being slowed down. At that time a new model file can be created and imported with this option.

### Unit

Specifies the unit that the STL file was created in. The STL file does not contain any unit information, only relative locations. Therefore, the proper unit must be selected here so all the relative locations in the STL will correspond to the correct absolute dimension in COOL3D. This is a simple scaling operation so if the imported shape does not appear correct, the STL file can easily be imported again with a different unit selection. The unit choices include:

- **cm**
- **mm**
- **in**
- **ft**
- **micron**
- **km**
- **mi**
- **dm**





## Import STL as Cross Sections - Import a shell from an STL file using cross sections

This operation is used to import an STL file containing a shell into GEM3D. This will only work to import the shell. Any interior components (like baffles and pipes) will be ignored and must be built manually in GEM3D. Before this operation can be done, the STL file should be generated from the CAD data following these guidelines:

- The coordinate system of the STL file should be chosen so that the main flow direction is aligned along a coordinate axis (X,Y, or Z).
- The surface of the shell in the STL file should be solid. There should be no holes in the shell.
- Remove internal components (pipes, baffles, etc.) if possible. This is not mandatory but it will decrease the time necessary to run the conversion. Also, the STL importer will ignore all surfaces except the outermost ones, so they will not be imported into GEM3D anyway.
- The STL file should not contain wall thickness. In other words, it should represent a single inside surface. This is not necessary, but the STL importer will use the outermost surface, which is not the correct size since the flow will see the internal surface.
- Remove external appendages that do not affect the volume of the interior of the shell (such as a solid boss that extends out from the shell).

## STL Convert

---

<b>STL File</b>	Specifies the name of the STL file containing the shell to be imported.
<b>Cross Sections</b>	Specifies the number of cross sections that will be taken between the shell ends. The number of cross sections requested must be between 4 and 200. The cross sections will be taken perpendicular to the <b>Major Axis</b> specified below and will be evenly spaced along the <b>Major Axis</b> . The number of cross sections chosen should be sufficiently large to accurately define the shape of the shell.
<b>Vertices</b>	Specifies the number of radial points to be used on all cross sections to define their shapes. These points will be spaced in equal angular increments around the center of the cross-section (e.g. if 90 vertices are chosen, there will be one point every 4 degrees). The number chosen should be sufficiently large to accurately define the shape of any cross section of the shell. The number of vertices requested must be between 10 and 200. The default value is 30.
<b>Major Axis</b>	Specifies the coordinate direction that is aligned with the major axis (main direction of flow) of the shell in the STL file. This axis will maintain its direction in the GEM3D model file. If the major axis is not known, select "def". In the rare case that the muffler is wider than it is long, "def" will not align the shell properly and the appropriate axis (X, Y, or Z) must be selected. The coordinate direction choices include: <ul style="list-style-type: none"> <li>• <b>def</b> indicates the major axis will be the direction of the longest shell dimension (length, width, or height).</li> </ul>



- **X-axis** indicates the major axis will be the X-axis direction in the STL file.
- **Y-axis** indicates the major axis will be the Y-axis direction in the STL file.
- **Z-axis** indicates the major axis will be the Z-axis direction in the STL file.

### Volume

Specifies the internal volume of the STL file (shell) in the units given by the **Unit** drop-down below. This value will be used to scale all of the dimensions of the shell so that the overall volume is correct. This scaling has the main purpose of making the internal volume of the shell accurate despite any approximations in describing the shell shape. If the **Volume** is set to "def" or "0", no scaling of the dimensions from the STL file will occur. In this case the volume will be determined from the STL file assuming that it was created in the units given by the **Unit** drop-down.

### Unit

Specifies the unit that the STL file was created in. If a **Volume** of 0 is specified above, then this is the unit that the STL importer will assume the STL file was created with. If a non-zero **Volume** is specified above, then this specifies the unit for that specified **Volume**. The unit choices include:

- **m<sup>3</sup>**
- **cm<sup>3</sup>**
- **mm<sup>3</sup>**
- **in<sup>3</sup>**
- **ft<sup>3</sup>**
- **Liter**
- **US-gallon**

### Tolerance

The tolerance controls the numerical scheme used in the STL importer. The value must be between "0" and "1". The default value of 0.01 (1.0%) will be sufficient for most STL imports. If the import routine fails, the above attributes should be varied to see if the STL file can be successfully imported. If the import routine still fails, increasing the tolerance may solve the problem and allow a successful import. This should only be done as a last resort since a larger tolerance can cause other problems for calculations after the shape is imported.

It is important to understand how the STL importer handles the ends of the muffler shell. The STL importer cannot determine the geometry at either end of the muffler shell, since the cross section is ill-defined, and can range from a single point to solid cross section. To get around this difficulty, the shape at either end is assumed to be identical to the nearest cross-section inside the shell.

After the shape of each end is imposed, the total volume of the muffler shell is calculated. If the user entered a non-zero value for the shell volume, all dimensions of the shell will be scaled so that the calculated volume equals the user-entered value. After importing the file into the pre-processor, it is important to inspect the various cross sections, especially the two end sections. It may be necessary to change the shape of the end sections to more accurately reflect the true geometry. In cases where there is



an extreme change in the cross-sectional shape near either end of the muffler, it may be more convenient to set the user-entered volume to "0". This way, no scaling of the dimensions will occur.



## Import IGES - Import an entire IGES file

This operation is used to import the entire contents of an IGES file into GEM3D. During the import routine, all the shapes in the file are determined and used to create the shape in GEM3D. Each section of the file (see the **Split by connection** option below) will be imported as a '[GEMMeshShape](#)' component. If the model file is empty, then the graphical view will automatically be reset to show the imported shape. If the model file is not empty (contains already existing components), then existing graphical view will be left alone. This can result in the imported shape not being shown because its absolute location is off the screen (see the **Center at Origin** option below). The '[GEMMeshShape](#)' components cannot directly be discretized into flow components for use in GT-SUITE, so after the import operation they must be divided and converted into GEM3D components for discretization. For details about dividing and converting the mesh shapes, see the operations on the [Convert Menu](#).

The ability to divide and convert imported shapes depends very strongly on the quality and nature of the geometry in the IGES file. The following guidelines should be followed to ensure the dividing and converting process is robust and consistent. Even if the guidelines are not followed, the mesh shapes may still be able to be handled (divided and converted), but the chance of success goes down.

- All holes in surfaces should represent flow ports. Holes in surfaces that will not be used as a flow port could potentially cause errors during conversion. For more information about holes, see the **Close Surface Holes** import option below.
- Ideally, the IGES file should not contain wall thickness. In other words, it should represent a single inside surface. If the shapes do contain wall thickness, the outer wall can usually be removed with the [Separate by Curves](#) operation or the [Marking](#) operation.
- The IGES file should represent the inside surface (that the flow sees) of the final assembly (or single component). In other words, if multiple mesh shapes need to be combined together to create a component, the chance of success goes down. For example, a pipe component should be created from a single mesh shape, and not from 2 mesh shapes that may represent two halves of the pipe. The Patch Mesh Ports operation will be able to help, as well as other [Convert Menu](#) operations. The [GEMMeshShell - Mesh Shell](#) component also will usually work in these situations.

## Import Options

---

### Simple Mode

When checked, a simple import algorithm is used. This method is very fast and works on large files, but is not very accurate at all. Therefore, this option should only be used if the import operation is failing, crashing, locking up, or running out of memory.

### Split by connection

When checked, the imported shape will contain multiple sections based on the connections in the file. This should be checked if the imported shape contains multiple separate sections, and these separate sections should be created as different components. When not checked, the imported file will be a single component.

### Center at Origin

When checked, the centroid of the imported component will be placed at the origin (0,0,0). This should be checked if the absolute location doesn't matter and it would be beneficial to have the entire model centered. When unchecked, the imported file will be placed exactly as it was when



the file was created. This should not be checked if the absolute location of the model is important and should be retained. If the absolute location is retained, then the imported shape will be placed at that exact location. If there are existing components in the model, then the current graphical view will be maintained, which could result in the imported shape being off the screen.

### **Close Surface Holes**

When checked, all holes that are on the surface of any shape in the imported file will be automatically closed (very similar to the [patching operation](#)). Normally this option can be left un-checked to preserve the exact nature of the geometry in the imported file. This can be checked for large volumes that need to be discretized like mufflers and air boxes to make the volume calculation and conversion routines more accurate. If the holes were not closed and the calculations are not accurate, it is usually possible to use the [patching operation](#) to close the necessary holes.

### **Import Resolution**

Specifies the resolution to be used while importing the IGES file into GEM3D. A value of "ign" or 1 will use the default resolution. Values less than 1 (minimum=0.1) will have a lower resolution and values greater than 1 (maximum=10) will have a higher resolution.

Generally, this option should be left as "ign" as the default resolution will function very well in GEM3D. A lower resolution can be used to reduce the model size and computational load. If GEM3D is very slow when importing with the default resolution, choosing a lower resolution will speed up the response while sacrificing accuracy of calculations. A higher resolution can be used to increase the accuracy of calculations. If the calculated dimensions are not precise enough, choosing a higher resolution will improve the accuracy of these calculations.



## Import ACIS - Import an entire ACIS file

This operation is used to import the entire contents of an ACIS file into GEM3D. During the import routine, all the shapes in the file are determined and used to create the shape in GEM3D. Each section of the file (see the **Split by connection** option below) will be imported as a '[GEMMeshShape](#)' component. If the model file is empty, then the graphical view will automatically be reset to show the imported shape. If the model file is not empty (contains already existing components), then existing graphical view will be left alone. This can result in the imported shape not being shown because its absolute location is off the screen (see the **Center at Origin** option below). The '[GEMMeshShape](#)' components cannot directly be discretized into flow components for use in GT-SUITE, so after the import operation they must be divided and converted into GEM3D components for discretization. For details about dividing and converting the mesh shapes, see the operations on the [Convert Menu](#) menu.

The ability to divide and convert imported shapes depends very strongly on the quality and nature of the geometry in the ACIS file. The following guidelines should be followed to ensure the dividing and converting process is robust and consistent. Even if the guidelines are not followed, the mesh shapes may still be able to be handled (divided and converted), but the chance of success goes down.

- All holes in surfaces should represent flow ports. Holes in surfaces that will not be used as a flow port could potentially cause errors during conversion. For more information about holes, see the **Close Surface Holes** import option below.
- The ACIS file should represent the inside surface (that the flow sees) of the final assembly (or single component). In other words, if multiple mesh shapes need to be combined together to create a component, the chance of success goes down. For example, a pipe component should be created from a single mesh shape, and not from 2 mesh shapes that may represent two halves of the pipe. In any situation where the ACIS file does not just represent the entire inside surface, the best solution is to use GT-SpaceClaim to extract the inner volume. It is also possible that the Patch Mesh Ports operation will be able to help, as well as other [Convert Menu](#) operations. Converting the solid to a mesh shape, and then to a [GEMMeshShell - Mesh Shell](#) component also will usually work in these situations.

This operation is also available from the command line, by using the "gem3d" command followed by the name of the ACIS file to be imported. The default options ("Split by connection" and "Import as solid model") will be used, other options are not available when importing from the command line.

---

## Import Options

### Split by connection

When checked, the imported shape will contain multiple sections based on the connections in the file. This should be checked if the imported shape contains multiple separate sections, and these separate sections should be created as different components. When not checked, the imported file will be a single component.

### Center at Origin

When checked, the centroid of the imported component will be placed at the origin (0,0,0). This should be checked if the absolute location doesn't matter and it would be beneficial to have the entire model centered. When unchecked, the imported file will be placed exactly as it was when the file was created. This should not be checked if the absolute location of the model is important and should be retained. If the absolute location



is retained, then the imported shape will be placed at that exact location. If there are existing components in the model, then the current graphical view will be maintained, which could result in the imported shape being off the screen.

### **Import Resolution**

Specifies the resolution to be used while importing the ACIS file into GEM3D. A value of "ign" or 1 will use the default resolution. Values less than 1 (minimum=0.1) will have a lower resolution and values greater than 1 (maximum=10) will have a higher resolution.

Generally, this option should be left as "ign" as the default resolution will function very well in GEM3D. A lower resolution can be used to reduce the model size and computational load. If GEM3D is very slow when importing with the default resolution, choosing a lower resolution will speed up the response while sacrificing accuracy of calculations. A higher resolution can be used to increase the accuracy of calculations. If the calculated dimensions are not precise enough, choosing a higher resolution will improve the accuracy of these calculations.



## Internal Volume - Calculates the internal volume of components or shapes

This operation is used to calculate the internal volume of components or shapes. To calculate the internal volume of a component or shape, first select the component or shape, then click on the Internal Volume operation (Dimension menu). The internal volume of multiple components or shapes can be calculated by selecting all the components or shapes and then selecting the Internal Volume operation.

The internal volume calculation works differently for GEM3D components, solid shapes ('[GEMSolidShape](#)'), and mesh shapes ('[GEMMeshShape](#)').

- GEM3D components - The volume calculated represents the volume contained inside the component. Since the GEM3D components have a very well defined geometry, the volume inside the component can be easily and very accurately calculated.
- Solid and mesh shapes - The volume calculated still represents the volume contained inside the component. However, due to the geometry of shapes, this volume calculation method is not as reliable as it is for the GEM3D components. This is a byproduct of the way the shape geometry is defined. Its shape is determined by a general surface, so GEM3D does not know that a particular surface is a wall (inner or outer), so the volume operation is not as well-defined.

### Internal Volume Calculation

<b>Name</b>	Displays the name.
<b>Type</b>	Displays the type of component or shape.
<b>Volume</b>	Displays the internal volume.
<b>Unit</b>	Specifies the unit in which to display the internal volume.
<b>Remove volume from shell occupied by internal components or features</b>	Toggle to remove internal volumes. When not checked, the calculated internal volume(s) will be for each component or shape by itself. When checked, the calculated internal volume(s) will include the subtraction of volume occupied by internal components (pipes, baffles, etc.). Therefore, the internal volume of a component will be smaller if it has internal components inside of it (like shells).
<b>Total volume</b>	Displays the total volume of all selected components or shapes. This will be the sum of all the individual internal volumes of the items above.





## **Length - Calculates the length of one or more components**

This operation is used to calculate the length of flow components along their flow path. This is especially useful for calculating an unknown length of a series of parts, some of which may contain bends in all three dimensions.

For certain components, the length calculation depends on which flow components are selected together. When measuring the length of a YSplit, the calculated length will be that of the branch connected to the adjacent selected flow component. Note that if only the YSplit is selected to measure length, a calculation of 0mm will result. An adjacent component must also be selected in order to determine which branch length will be measured.

---

### **Length Calculation**

---

<b>Name</b>	Displays the component or shape name.
<b>Type</b>	Displays the type of component or shape.
<b>Length</b>	Displays the length of the component or shape.
<b>Unit</b>	Specifies the unit in which to display the length.





## **Material Volume - Calculates the material volume of components or mesh shapes**

This operation is used to calculate the material volume of components or mesh shapes. The material volume is the volume occupied by the material used to create the component or mesh shape, effectively measuring the amount of material it would take to physically make the component or mesh shape. To calculate the material volume of a component or mesh shape, first select the component or mesh shape, then click on the Material Volume operation (on the Dimension menu). The material volume of multiple components or mesh shapes can be calculated by selecting all the components and then selecting the Material Volume operation.

---

### **Material Volume Calculation**

---

<b>Name</b>	Displays the name of the component or mesh shape for which the material volume was calculated.
<b>Type</b>	Displays the type of component or mesh shape for which the material volume was calculated.
<b>Volume</b>	Displays the material volume of the component or mesh shape.
<b>Unit</b>	Specifies the unit in which to display the material volume.
<b>Total volume</b>	Displays the total material volume of all selected components or mesh shapes. This will be the sum of all the individual values of the items above.





## Mesh Flowsplit Editor - Graphical editor to modify mesh flowsplit components

The mesh flowsplit editor allows the user to visualize and override the flowsplit angles, characteristic lengths, and expansion diameters of the mesh flowsplit ('[GEMFsplitGeneral](#)'). The editor also allows full graphical manipulation of the mesh flowsplit and its neighboring components in order to see it at various desired angles.

### Mesh Flowsplit Boundary Data

This section displays the Boundary Data tab of the '[GEMFsplitGeneral](#)' template. This allows access to the override values for the port angles, characteristic lengths, and expansion diameters so that they can be modified. Pressing the update view button at the bottom of the mesh flowsplit editor window will redraw the component in the graphical window with any changes that were made to the port data information. For more information about these attributes, please see the help for the '[GEMFsplitGeneral](#)' template.

### Visibility Table

This section is used to control the display of the port information and adjacent components.

**ID** - This corresponds to the column number of the port data information above. The ID row in the visibility table will control the port and neighbor visibility for the equivalent column number of the port data tab.

**Ports** - When checked, this option will visibly display the port angle, characteristic length, and expansion diameter of the mesh flowsplit component.

**Neighbor** - When checked, this option will visibly display the adjacent component connected to the associated port. This component will be displayed at a higher transparency level than in the modeling graphical display so that it does not interfere with the visibility of the current mesh flowsplit being edited. This option will be un-editable (yellow background) if the port is not connected to another component.

### Graphical Window

This section displays the current mesh flowsplit component, along with any other information based on the visibility selection above. This graphical window allows full graphical manipulation of the model including rotation, panning, and zooming. This allows the mesh flowsplit attribute values to be seen graphically on the mesh flowsplit component and its neighbors. This makes it easier to verify the calculated angles, lengths, and diameters. It also provides a graphical aid in determining override values if necessary. The graphical window contains the following graphical aids.

**Black** circle with a black arrow (perpendicular direction to port) = Expansion diameter

**Black** disks with a black arrow between them (parallel direction to port) = Characteristic length

**Green** disk = Neighbor equivalent diameter (for pipe components)

**Red** disk = Neighbor expansion diameter (for flowsplit components)

**Magenta** disk = Connection diameter (if a connection is present)



## **Model View Layout - Controls the layout of the model in the display window**

This window allows the user to modify the views that are displayed in the graphical window. Here the number of views, individual views, and layout can be configured for both the model tab and preview tab. A preview of the resulting graphical layout is shown at the bottom of the window.

**Note:** This preview does not include the display of the actual model, simply the layout.

### **Panel Layout for Model / Preview**

---

<b>Number of display panels</b>	Specifies the number of view panels to be displayed in the graphical window. Up to 4 views may be used.
<b>Display panels layout</b>	Specifies the orientation of the display panels.
<b>Panel configuration</b>	Specifies what view each of the display panels will show. Each display panel can show any view, including duplicates.



## Parallel Baffle to Baffle - Creates a baffle parallel to an existing baffle

This operation is used to create a baffle parallel to an existing baffle.

- 1) To create a parallel baffle, first select an existing baffle, then click the Parallel Baffle to Baffle operation. This will open the parallel baffle dialog window.
- 2) Specify the distance from the selected baffle where the parallel baffle will be created.

### Parallel creation

---

**Distance (+/-)** Specifies the distance from the selected baffle where the parallel baffle will be created. A positive distance will be in the direction of the normal vector to the selected baffle. A negative distance will be in the opposite direction.

- 4) Click OK to finish the operation. The parallel baffle will be created using the specified distance. All other properties of the baffle will be identical to the selected baffle.





## Parallel Datum Plane - Creates a datum plane parallel to an existing datum plane

This operation is used to create a datum plane parallel to an existing datum plane.

- 1) To create a parallel datum plane, first select an existing datum plane, then click the Parallel Datum Plane operation. This will open the parallel datum plane dialog window.
- 2) Specify the distance from the selected datum plane where the parallel datum plane will be created. A distance value of 0 would result in the new datum plane being created in the exact same location as the selected datum plane.

### Parallel creation

---

**Distance (+/-)**

Specifies the distance from the selected datum plane where the parallel datum plane will be created. A positive distance will be in the direction of the normal vector to the selected datum plane. A negative distance will be in the opposite direction.

- 4) Click OK to finish the operation. The parallel datum plane will be created using the specified distance. All other properties of the datum plane will be identical to the selected datum plane.





## Polygon Vertices - Allows creation and editing of custom cross section shape by coordinate points

This table is used to edit or create polygon vertices in order to create a custom cross section. An X and Y coordinate position is entered in each row to create the polygon vertices. By right clicking in a cell, the user can append, insert, or delete rows. Rows that are left blank will be ignored.

**Note:** The polygon's vertices must be entered in the order in which they should be connected. For instance, a line will be created between the vertices entered in row 1 and row 2 to create the first polygon edge. Next, the second polygon edge will be created between the vertices in row 2 and 3, and so on. The vertices order determines how the polygons' edges will be built and therefore what the polygon shape will be.

### Table

X	X position of vertex point
Y	Y position of vertex point

**Unit:** Drop down menu specifying the current unit for the points of the cross section. Choices include m, cm, mm, in, ft, micron, and dm.

**Resize table:** This button allows the user to modify the number or rows in the table.



## **Print - Prints the graphical display**

This command is used to print the graphical views.

### **Print active view**

Prints the currently active view.

### **Print all views**

Prints all views in the graphical display. Each view will be printed individually on its own page.

### **Print all views on one page**

Prints all views in the graphical display on a single page. This will result in a printed page that looks exactly like the display window.







## **Print Preview - Previews the graphical display**

This command is used to print the graphical views.

### **Preview active view**

Previews the currently active view.

### **Preview all views**

Previews all views in the graphical display. Each view will be shown individually on its own page.

### **Preview all views on one page**

Previews all views in the graphical display on a single page. This will result in a page that looks exactly like the display window.





## Pipe Normal Plane Control Dialog – Location of a pipe normal plane

This window provides some detailed geometrical information about a “Pipe Normal” cutting plane or datum plane. The top section of the window contains a graph of the **Effective Diameter** and **Angle** as a function of the **Distance** along the component. The bottom section contains real-time counters giving the numerical values at the current location of the local cutting plane. The local cutting plane can be controlled graphically using the mouse, from the graph using the triangle at the top, or by the keyboard arrow keys (used to make small move increments).

**Note:** The local cutting plane must first determine the centerline of the component. This process is more accurate if the initial cutting plane location is placed along a straight section, preferably near the middle (i.e. away from any bends or surface features). If the graph shows unusual values, pressing cancel in this window and choosing a different initial position may help.

---

### Pipe Normal Plane

---

<b>Distance</b>	Shows the current distance of the plane along the component.
<b>Effective Diameter</b>	Shows the current effective diameter of the plane’s intersection with the component.
<b>Angle</b>	Shows the relative angle between the current orientation of the plane and the orientation at the end of the component. If this angle is constant along the length of a shape, then that section is straight. If this angle is changing, then that section is bending. The point at which this angle starts or stops changing gives the location of the start or end of bent sections.

**Clip:** This button is only available when the function was launched from a “Cutting Plane” button. It will separate the component at the plane location into multiple components.

**Save as Datum:** This button will store a datum plane at the current plane location (without separating the component into multiple components).



### Set Rotation Point - Sets the anchor point for graphical rotations

This command is used to set the anchor point for graphical rotation operations. This anchor point will be used when conducting graphical rotations in the screen XY plane (left mouse button + mouse drag) and about the screen Z axis (ctrl key + left mouse button + mouse drag). The anchor point is stored as an application property.

<b>Global origin (0,0,0)</b>	Sets the rotation anchor point to the origin. The origin is the point with global coordinates of X=0, Y=0, and Z=0.
<b>Global centroid</b>	Sets the rotation anchor point to the centroid (center of volume) of the model. When this option is selected, the centroid of the entire model is calculated and set as the anchor point. This option is the default choice.
<b>Component's centroid</b>	Sets the rotation anchor point to the centroid of any selected component. When this option is used, any component can be selected from the graphical window. The currently selected component's name will be given and its centroid point (X, Y, and Z location) will be set as the rotation anchor point. The particular centroid point will be remembered and used as the rotation anchor point when this window is closed.
<b>Select control point</b>	Allows the rotation anchor point to be set by the coordinates of a selected control point or the absolute coordinates specified by the <b>X</b> , <b>Y</b> , and <b>Z</b> attributes below. When this option is selected, any control point in the model may be selected to be set as the rotation anchor point. When control points are selected, their coordinates will be filled in for the <b>X</b> , <b>Y</b> , and <b>Z</b> attributes below. This allows the coordinates to be checked and/or modified. The <b>X</b> , <b>Y</b> , and <b>Z</b> attributes may also be directly specified to set the rotation anchor point. The specified coordinates are in the <a href="#">global coordinate system</a> .
<b>X</b>	Absolute X coordinate in the <a href="#">global coordinate system</a> to be set as the X value of the rotation anchor point. This attribute will always update based on the current selection of the rotation anchor point. This attribute may only be manually edited when the <b>Select Control Point</b> option is enabled above.
<b>Y</b>	Absolute Y coordinate in the <a href="#">global coordinate system</a> to be set as the Y value of the rotation anchor point. This attribute will always update based on the current selection of the rotation anchor point. This attribute may only be manually edited when the <b>Select Control Point</b> option is enabled above.
<b>Z</b>	Absolute Z coordinate in the <a href="#">global coordinate system</a> to be set as the Z value of the rotation anchor point. This attribute will always update based on the current selection of the rotation anchor point. This attribute may only be manually edited when the <b>Select Control Point</b> option is enabled above.



## Surface Area - Calculates the internal surface area of components or shapes

This operation is used to calculate the internal surface area of components or shapes. The surface area is the internal flow area that is used for friction and heat transfer calculations. To calculate the surface area, first select the component or shape, then click on the Surface Area operation (on the Dimension menu). The surface area of multiple components or shapes can be calculated by selecting all the components and then selecting the Surface Area operation.

If the component or shape being measured includes perforations, the surface area calculation will represent the internal flow area of the component minus the area taken up by the perforations.

### Surface Area Calculation

---

<b>Name</b>	Displays the name of the component or mesh shape for which the surface area was calculated.
<b>Type</b>	Displays the type of component or mesh shape for which the surface area was calculated.
<b>Area</b>	Displays the surface area of the component or mesh shape.
<b>Unit</b>	Specifies the unit in which to display the surface area.
<b>Total area</b>	Displays the total surface area of all selected components or mesh shapes. This will be the sum of all the individual values of the items above.



## File → Options → General - Contains general options and preferences specific to the application

This tab contains general options that can be used to control how the GEM3D application operates.

---

### General

---

<b>Auto display toolboxes</b>	When this option is checked, GEM3D will automatically open and position all <a href="#">toolboxes</a> that were open the last time the application was closed.
<b>Icon only on button</b>	When this option is checked, GEM3D will only display the icons of operations in the toolboxes. When un-checked, GEM3D will display the icon and the name of the operation.
<b>Warn before deleting mesh or solid shapes</b>	When checked, a warning message will be displayed any time mesh and/or solid shapes are deleted. This is simply a confirmation that the shapes should really be deleted. When not checked, the shapes will be deleted without any further confirmation from the user. This is an application toggle. It can be turned off from the warning message dialog window. To turn it on again, it must be done from File → Options.
<b>Do not highlight component when mouse cursor is on it</b>	When checked, components and features will not be highlighted with a yellow color when the mouse cursor is over them. This option will improve the speed of graphical operations in GEM3D. When not checked, components and features will be highlighted in a yellow color to denote which component/feature would be selected if the mouse button were pressed. This option will display exactly what would be selected (leaving nothing to chance) and allow a preview of selection that may prevent accidental operations.
<b>Highlight attributes that affect modeling</b>	When checked, this option will cause the name of all attributes that affect the geometrical modeling to be displayed in blue (instead of the normal black text).
<b>Import dialog center at origin</b>	When checked, this option will always have the center at origin option checked during an <a href="#">Import STL as Surface</a> operation.
<b>Re-normalize directions after pressing 'Apply' or 'Ok'</b>	When checked, all direction attributes will be automatically re-normalized after pressing "Apply" or "OK".
<b>Number of significant figures displayed</b>	The number specified in this dropdown will set the number of significant figures to be displayed for all attributes. By default, seven significant figures are displayed. The value selected here will not affect the discretization of the model. If a value has more significant figures than what is specified for this option, then all values will be stored properly.
<b>Default Transparency Percentage [0-90]</b>	Indicates the default transparency level used when an individual component has a <b>Transparency Percent</b> of "def". 0 indicates opaque (solid) and 90 indicates almost completely transparent.




## File → Options → Favorites - Contains options regarding favorite folders and applications

This tab contains options regarding favorite folders and applications.

---

### Favorites

---

<b>Default Template Library</b>	Default template library to be used by GEM3D.
<b>Default Editor</b>	Default text editor used by the application. When this is left blank, the application's default text editor is Notepad on Windows.
<b>SpaceClaim Path</b>	This is the location of the GT-SpaceClaim executable. We typically launch GT-SpaceClaim by reading a registry key that contains the GT-SpaceClaim installation path, but this preference will be used if the registry key is invalid or cannot be read.
<b>Favorite Folder</b>	 Specifies the application's favorite folder. When saving or opening a file, the Favorite Folder can quickly and easily be accessed by clicking on the icon that appears in the upper-right corner of the dialog box.



## File → Options → Save - Contains options and preferences regarding saving models

This tab contains options that can be used to control how GEM3D saves models.

---

### Save

---

#### AutoSave every \_\_\_\_ minutes

When this option is checked, each open \*.gem file will automatically be saved at an interval specified by the user. The file will be saved in the same directory with the same name as the original file except that an additional extension of *.autosave* will be appended to the filename. To recover data with AutoSave, simply open the *\*.autosave* file, at which point the user will be prompted to resave it under a different name. A prompt will automatically appear any time that a \*.gem file is opened when there is an analogous *.autosave* version with a newer time and date in the same directory. Beware that when recovering an AutoSave file, do not open the original file prior to or simultaneously with the AutoSave file, because the AutoSave file can be automatically overwritten or erased when the original file is successfully closed.

#### Issue warning on exit

When this option is checked, GEM3D will request confirmation whenever the user attempts to exit the application.



## **File → Options → Default Units - Contains the default units preferences**

This tab contains the preferences for the default units to be used in the GEM3D application.

---

### **Default Units**

---

<b>Length</b>	Default length unit used by the application.
<b>Area</b>	Default area unit used by the application.
<b>Volume</b>	Default volume unit used by the application.
<b>Angle</b>	Default angle unit used by the application.
<b>Inverse area</b>	Default inverse area unit used by the application.





## **File → Options → Default Colors - Contains the default colors preferences**

This tab contains the preferences for the default colors to be used in the GEM3D application.

---

### **Default Colors**

---

<b>Default Component/Feature Display Color</b>	Specifies the default color to be used for creation of new components and features.
<b>Background Color</b>	Specifies the background color of the canvas (graphical window).
<b>Highlight Color</b>	Specifies the color that components and features will be displayed when highlighted.
<b>Select Color</b>	Specifies the color that components and features will be displayed when selected.
<b>Wheel Color</b>	Specifies the color of the graphical operation wheel located in the upper right corner of the canvas (graphical window).



## File → Options → Conversion - Contains the conversion preferences

This tab contains the preferences for the conversion of mesh or solid shapes into components.

### Conversion

#### Allow Mesh Deconversion

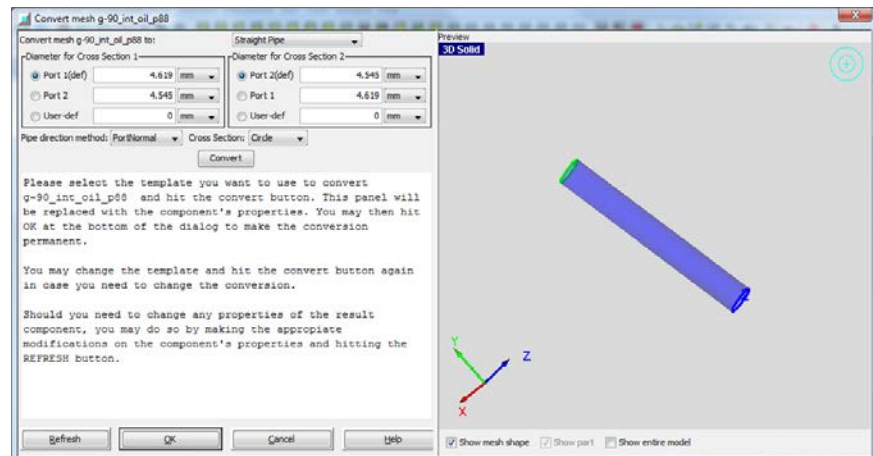
When checked, this option will allow a GEM3D component converted from a mesh shape to be de-converted back into its original mesh shape. The advantage to this option is the ability to go directly back to the mesh shape if the conversion options need to be changed or the mesh shape modified. The disadvantage is a performance penalty as a significant amount of extra information must be saved in the model file for each shape that can be de-converted. Due to this performance penalty, it is recommended not to use this option unless a specific use is needed. It is always possible to import the STL file again to get the mesh shape.

#### Default Color for Converted Component

Specifies the default color to be used for a component that is created by [converting](#) a mesh shape.

#### Number of digits after decimal point displayed

The number specified in this dropdown will set the number of digits to display after the decimal point when converting mesh shapes into pipes, flowsplits, and shells. This option will apply to all attributes including diameter, length, volume, surface area, etc. By default, three digits are displayed after the decimal, as shown in the example conversion dialog below.



#### Default Wall Thickness

Specifies a default value for the wall thickness of the converted component.

#### Mesh Preview Transparency Percent

Specifies the transparency percentage of the mesh shape in the preview window.

#### Default Conversion Part Names

Determines whether the prefilled name of a solid or mesh shape being converted to a GEM component will be set as:

- None (Empty)
- Default Names – GEM hardwired name per component template



- Original Shape Names – The name of the original solid or mesh shape will be re-used for the converted part

**Method for determining  
User Characteristic  
Dimension for  
GEMFSplitGeneral Ports**

For GEMFSplitGeneral components that were converted from Solid or Mesh shapes, where the **Flowsplit HTC Calculation Method** is set to “Method #2”, and the **Characteristic Dimension for Port HTC** is set to “User\_Dimension”, the following options are available for GEM to set the **User Characteristic Dimension** in Boundary Data Folder

- **Equivalent Diameter (based on Area):** GEM will calculate the equivalent diameter of a circle that has the same area that was measured at the flowsplit port, and enter that value for the User Characteristic Dimension.
- **Hydraulic Diameter:** GEM will calculate the hydraulic diameter at the flowsplit port, and enter that value for the User Characteristic Dimension.



## File → Options → Discretization - Contains the discretization preferences

This tab contains the preferences for the discretization of components into a model file.

---

### General

---

#### Warn before overwriting gtm file

When checked, this option will have GEM3D issue a warning before overwriting an existing .gtm file with the same name as the currently specified output file name. This can prevent an exported model file from overwriting a potentially useful model file. When not checked, GEM3D will automatically overwrite the existing file with the currently exported model file. If a previous model file with the same name does not exist in the specified save directory, then this option does nothing.

#### Discretize PipeXYZPoints into individual parts

When checked, GEM3D will create individual parts for each straight and bent section of any '[GEMPipeXYZPoints](#)' component. Normally this component is discretized into the '[PipeTable](#)' object to make the project map cleaner and easier to navigate. This option can be used if individual parts are desired to easily identify where straight and bend sections start and/or end.

#### Model License Type

Specifies the license that will be used to run the exported GT-ISE model file (.gtm). Note that the license type chosen here can always be changed in GT-ISE if necessary.

#### Preview options

Specifies the default application property for the type of preview option used to display the discretization preview in the graphical window. The discretization preview will discretize any shells in the model and figure out how many flowsplits are needed. Then, it will display all the flowsplits needed in the graphical window according to this preview option. The discretization preview is only done in a 3D view of the model. Therefore, if no 3D view currently exists, then the preview button will be disabled (grayed out). This value can be overridden by the same option in the [Export .gtm](#) operation. Available preview options include:

- **Show cubed** displays each flowsplit as a separate cube. The size of the cube is proportional to the discretization length of the shell. All the cubes in a particular chamber will be displayed as the same color.
- **Show cube with volume** displays each flowsplit as a separate cube with different brightness. A lighter colored cube represented a larger percentage of the cube's volume being inside the shell. A darker cube represents a smaller percentage.
- **Show cube with mesh line** displays both the cubes and the mesh lines.
- **Show mesh line** displays lines that represent the edges of each flowsplit.
- **Show mesh face** displays planes (faces) that represent the division between flowsplits.



	<ul style="list-style-type: none"><li>• <b>Show mesh face and lined</b> displays both the mesh faces and lines.</li></ul>
<b>Cube transparency level (%)</b>	Specifies the default application property for the transparency level used when drawing the preview cubes (when using the cube preview option). 0 indicates opaque (solid) and 90 indicates almost completely transparent. This value can be overridden by the same option in the <a href="#">Export .gtm</a> operation.
<b>Object Names - Prefix</b>	Specifies a user defined text string that will be added to the beginning of all object names that are created during the discretization process. This should be used when multiple GEM3D models with similar component names will be used as subassemblies in the same GT-SUITE model file to prevent duplicate object names. This prefix is limited to a maximum of 3 characters. If this prefix results in creation of an object name longer than the maximum allowed in a model file (20 characters), an error will be given and the name will be truncated from the end. This box can be left blank and no prefix will be added.
<b>Object Names - Suffix</b>	Specifies a user defined text string that will be added to the end of all object names that are created during the discretization process. This should be used when multiple GEM3D models with similar component names will be used as subassemblies in the same GT-SUITE model file to prevent duplicate object names. This suffix is limited to a maximum of 3 characters. If this suffix results in creation of an object name longer than the maximum allowed in a model file (20 characters), an error will be given and the suffix will be omitted or truncated. This box can be left blank and no suffix will be added.
<b>Part Names - Prefix</b>	Specifies a user defined text string that will be added to the beginning of all part names that are created during the discretization process. This should be used when multiple GEM3D models with similar component names will be used as subassemblies in the same GT-SUITE model file to prevent duplicate part names. This prefix is limited to a maximum of 3 characters. If this prefix results in creation of a part name longer than the maximum allowed in a model file (20 characters), an error will be given and the name will be truncated from the end. This box can be left blank and no prefix will be added.
<b>Part Names - Suffix</b>	Specifies a user defined text string that will be added to the end of all part names that are created during the discretization process. This should be used when multiple GEM3D models with similar component names will be used as subassemblies in the same GT-SUITE model file to prevent duplicate part names. This suffix is limited to a maximum of 3 characters. If this suffix results in creation of a part name longer than the maximum allowed in a model file (20 characters), an error will be given and the suffix will be omitted or truncated. This box can be left blank and no suffix will be added.
<b>Gravity/Acceleration Options</b>	<p>The two options are:</p> <ul style="list-style-type: none"><li>• <b>Consider Gravity:</b> When this option is selected, the user needs to specify the vector describing the direction that gravity will act. Only a unit vector is needed to describe the direction of gravity. This is</li></ul>



necessary to automatically calculate the correct elevation change for pipes.

- **3D Acceleration:** When this option is selected, the pipe parts with "Pipe Elevation Change or 3D Acceleration Object" attribute set to "def" will automatically get the 'PipeMotion3DBM' reference object created in the exported .gtm file where the 'PipeMotion3DBM' will have the inlet and outlet coordinates automatically obtained during discretization. Also, a 'BodyMotion' part will be created automatically in the exported .gtm file.

### Pipe Elevation Measurement Interval

The measurement interval used to determine the elevation change profile of a pipe along the flow length. If this attribute is unchecked, the elevation change of a pipe will be the difference between the outlet and inlet ports.

Note that this option is available only when the "Gravity/Acceleration Options" is set to **Consider Gravity**.



## Translation - Translates a component

This operation is used to translate a component a specified distance in the X, Y, and/or Z directions. This can be done in 2 ways; a) by specifying the exact distance to move in the dialog window or b) moving the component graphically with the mouse.

- 1) To begin either translation method, the component must first be selected.
- 2) Then click on the Translation operation. This will open the Translation dialog window and show a graphical axis on the component.
- 3a) Specify the distance to translate the component in the dialog window. The translation operation makes the current location of the selected component the 0 point (X, Y, and Z all equal 0). Therefore, the distance numbers specified in the dialog window are relative to the 0 point, or the original location of the component.
- 3b) Click and hold on any arrow (X, Y, or Z) of the graphical axis with the left mouse button and move the mouse to translate the component in that direction. The dialog window will automatically be updated and applied with the correct coordinates that the component was moved. To return the component to the original location, enter (0,0,0) in the dialog window and click Apply.

### Translate component

- |          |  |
|----------|--|
| <b>X</b> | Specifies the distance to translate the component in the X direction. This is relative to the original location of the component before the translate operation was started. |
| <b>Y</b> | Specifies the distance to translate the component in the Y direction. This is relative to the original location of the component before the translate operation was started. |
| <b>Z</b> | Specifies the distance to translate the component in the Z direction. This is relative to the original location of the component before the translate operation was started. |
- 4) Pressing OK will save all changes made and stop the translate operations. Pressing Close will stop the translate operation without applying any changes in the dialog window (any previous changes that were already applied, including graphical movements, will still be saved).





## View Model Sectioning - Sections the model to view interior component

This operation is used to place a temporary sectioning plane on the model so that interior components and features can be viewed. This is similar to a cross sectional view of a drawing. This sectioning plane can also be used during a discretization preview operation to be able see the interior flowsplits for shells.

### Sectioning

#### Orientation

Radio button that specifies which axis to slide the sectioning plane along. The sectioning plane will be oriented normal to the axis direction. Available orientations include:

- **X Axis** Specifies that the sectioning plane be placed normal to the global X axis.
- **Y Axis** Specifies that the sectioning plane be placed normal to the global Y axis.
- **Z Axis** Specifies that the sectioning plane be placed normal to the global Z axis.
- **Custom** Specifies that the sectioning plane be placed in a user-defined orientation. The sectioning plane can be rotated by clicking and dragging on either the head or tail of the arrow in the graphical window. It can also be translated by clicking and dragging anywhere on the plane. Once the sectioning plane is rotated off one of the normal axis directions, it will automatically become a custom orientation. The sectioning plane can only be oriented in the graphical window if the plane and arrow are visible (see the **Hide Arrow & Plane** option below).

Also shows a real-time counter of the current X, Y, and Z orientation of the normal vector of the sectioning plane in the default unit.

#### Rotate X

Slider bar that will rotate the sectioning plane about the X axis (in the YZ plane). Clicking outside of the control window will reset the position of the slider. The plane will be rotated by a specified amount given by the setting of the **Rotate plane** option below.

#### Rotate Y

Slider bar that will rotate the sectioning plane about the Y axis (in the XZ plane). Clicking outside of the control window will reset the position of the slider. The plane will be rotated by a specified amount given by the setting of the **Rotate plane** option below.

#### Rotate Z

Slider bar that will rotate the sectioning plane about the Z axis (in the XY plane). Clicking outside of the control window will reset the position of the slider. The plane will be rotated by a specified amount given by the setting of the **Rotate plane** option below.

#### Location

Shows a real-time counter of the current X, Y, and Z location of the center of the sectioning plane in the default unit (the ball of the arrow where it intersects the cutting plane).

#### Translate



Slider bar that will translate the cutting plane along the current





orientation. Clicking outside of the control window will reset the position of the slider. The plane will be translated by a specified amount given by the setting of the Translate plane option below.

**Translate plane**

Drop down option that determines how far the sectioning plane translates with each move of the **Translate** slider above. The amount is given in units since the absolute amount will depend on the current default unit selection of the length. Allowed values range from 0.1 to 5.

**Rotate plane**

Drop down option that determines how far the sectioning plane rotates with each move of the **Rotate** sliders above. Allowed values range from 1 to 10.

**Pos:** This option indicates that the part of the model on the positive side of the sectioning plane (normal direction) will be graphically removed to section the model.

**Neg:** This option indicates that the part of the model on the negative side of the sectioning plane (opposite of the normal direction) will be graphically removed to section the model.

**Hide Arrow & Plane:** Checking this option hides the sectioning plane and control arrow from the graphical window for display purposes. With the sectioning plane and arrow hidden, the sectioning plane cannot be oriented in the graphical window, but still can be oriented and translated using the controls in this dialog window. Un-checking his option will show the sectioning and control arrow again.



## **CHAPTER 4: GEM3D Templates**

The templates section contains information on each of the templates used in GEM3D. A description of each template is given as well as details about each attribute. These descriptions are the same text that can be found in the context help for each template. The context help can be viewed while using GEM3D by clicking the button in the upper left corner of each template. This button will have an image of the template as well as small question mark symbol.





## ActuatorConn3D - Actuator Connection

This template is used to actuate quantities of a flow part from control components part in the discretized .gtm model. The discretized model will have a 'ActuatorConn' and 'Gain' setup to automatically connect to other control components in the model.



### Main

#### Subassembly Port Number

Specifies the port ID number to be used for the external subassembly connection that is created during the discretization routine and connected to the actuator through a gain (see image above). This value will get copied directly into the **Corresponding Subassembly Port #** attribute in the 'SubAssExternalConn' object that is created in the GT-SUITE model file. This value can be any positive integer. If "def" is entered a number will be automatically assigned.

#### Signal Quantity to Actuate

Specifies the signal link ID number that corresponds to the quantity to be actuated. The value selector (right-click) can be used to select the actuated quantity from a list of allowed quantities.

#### Positioning for FlowSplit\* or Pipe\* (GEM3D Only)

#### ☉Coordinate System

Specifies the reference coordinate system used for specifying the location of the actuator. Choices include:

- **local** indicates that the location entered below will be measured from the local origin of the parent component.
- **global** indicates that the location entered below will be measured from the global origin.

#### (☉) Location X

Specifies the X location of the actuator in the local or [Global Coordinate System](#).

#### (☉) Location Y

Specifies the Y location of the actuator in the local or [Global Coordinate System](#).



**(☉) Location Z**

Specifies the Z location of the actuator in the local or [Global Coordinate System](#).

**☉Normalized Actuator  
Location**

**For Pipe\* Parts:** This attribute is used to specify a normalized axial location in 'Pipe\*' components at which the desired quantity is to be actuated (because there may be several discretized volumes within each 'Pipe\*' part). The normalized location is a value between 0 and 1 with "0" defining the end of the pipe at port 1 (the inlet) and "1" defining the end of the pipe at port 2 (the outlet). Setting this attribute to "ign" will cause the selected quantity to be sensed at a normalized location of 0.

Note: The normalized location for almost all actuated quantities is irrelevant (not used). There are only a few places where this attribute is useful, so it is very common to leave this attribute set to "ign". For the exceptions, see the context help for the 'ActuatorConn' template in GT-ISE.

---

**Visual**

---

**Display Color**

Indicates the color used when drawing the actuator. The color choices include:

- **Red**
- **Blue**
- **Dark Blue**
- **Green**
- **Copper**
- **Gold**
- **Grey**
- **Black**

**Show Actuator Label**

If checked, this will display the component name on the canvas.





## **ControlLine - Control Line**

This template is used to represent a control line. A control line can be created between any 2 control points. Control lines are mainly used for dimensioning.

### **Geometry**

---

#### **Length**

Length of the control line. A length of "def" will be set to twice the distance between the 2 points used to create the line.





## ControlPoint - Control Point

This template is used to represent the important points of components, referred to as control points. This usually includes vertex points of components and features. Control points are mainly used for dimensioning. Control points that are automatically created as the vertex points of components and features are not editable so the attributes will display with a yellow background. Control points created manually are editable.

### Geometry

---

<b>Location X</b>	Specifies the absolute X location of the control point.
<b>Location Y</b>	Specifies the absolute Y location of the control point.
<b>Location Z</b>	Specifies the absolute Z location of the control point.



## CSBiRadial - Bi-Radial Cross Section

This template is used to model a Bi-radial cross section shape. For more detailed information regarding this shape, see the drawing below. The anchor point of the Bi-radial cross section is the center of volume of the shape. Cross sections are used to build components like pipes and shells. Multiple cross sections of different shapes can be combined to create complex component shapes.

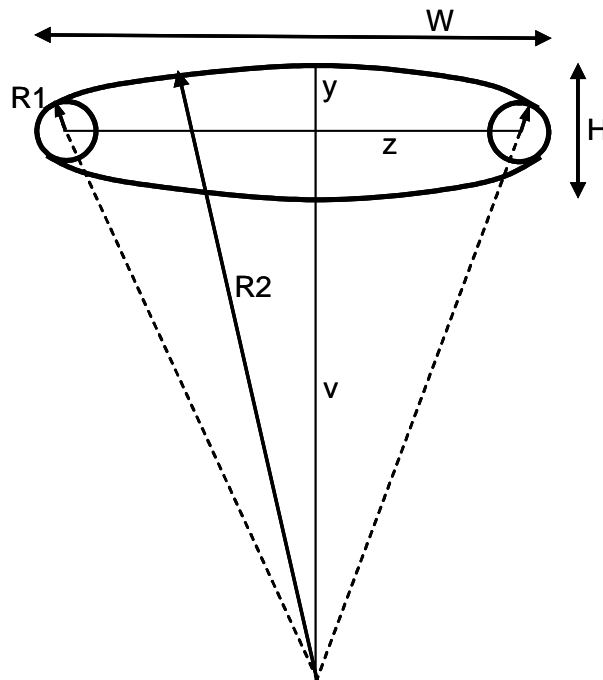
### Geometry

*The Bi-Radial shape is defined using the following 4 attributes. Any 1 of these attributes can be calculated if the other 3 are known. Therefore, any 1 of these attributes must be set to "def" while the other 3 are specified and its value will be calculated automatically.*

<b>Radius 1 (Small radius)</b>	Small radius to be used on the 2 sides of the shape ( $R_1$ ). "def" is allowed (see note above).
<b>Radius 2 (Large radius)</b>	Large radius to be used on the top and bottom of the shape ( $R_2$ ). "def" is allowed (see note above).
<b>Height</b>	Overall height of the shape ( $H$ ). "def" is allowed (see note above).
<b>Width</b>	Overall width of the shape ( $W$ ). "def" is allowed (see note above).

The Bi-Radial cross section shape is constructed from the drawing below and the following equations.

$$H = 2y \quad z = \frac{W}{2} - R_1 \quad y = R_2 - v \quad v = \sqrt{(R_2 - R_1)^2 - z^2}$$



Bi-Radial Cross Section Shape



## **CSCircle - Circular Cross Section**

This template is used to model a circular cross section. The anchor point of the circular cross section is the center of volume of the shape. Cross sections are used to build components like pipes and shells. Multiple cross sections of different shapes can be combined to create complex component shapes.

### **Geometry**

---

**Diameter**                      Diameter of the circular cross section.





## **CSCustom - User Defined Cross Section**

This template is used to model an arbitrarily shaped cross section that is defined by the user. This is accomplished by using the cross section editor. The cross section editor is used to create, modify, and save custom cross section shapes to be used to build components and features. For help using the cross section editor follow the link below.

[Cross Section Editor](#)



## **CSEllipse - Elliptical Cross Section**

This template is used to modal an elliptical cross section. The anchor point of the elliptical cross section is the center of volume of the shape. Cross sections are used to build components like pipes and shells. Multiple cross sections of different shapes can be combined to create complex component shapes.

### **Geometry**

---

<b>Major Diameter</b>	Specifies the major diameter of the ellipse.
<b>Minor Diameter</b>	Specifies the minor diameter of the ellipse.



## CSRect - Rectangular Cross Section

This template is used to model a rectangular cross section. The anchor point of the rectangular cross section is the center of volume of the shape. Cross sections are used to build components like pipes and shells. Multiple cross sections of different shapes can be combined to create complex component shapes.

### Geometry

---

<b>Height</b>	Specifies the height of the rectangle.
<b>Width</b>	Specifies the width of the rectangle.





## **CSRoundRect - Rounded Rectangular Cross Section**

This template is used to model a rectangular cross section with rounded corners. The anchor point of the rounded rectangular cross section is the center of volume of the shape. Cross sections are used to build components like pipes and shells. Multiple cross sections of different shapes can be combined to create complex component shapes.

### **Geometry**

---

<b>Height</b>	Specifies the height of the rounded rectangle.
<b>Width</b>	Specifies the width of the rounded rectangle.
<b>Radius</b>	Specifies the radius of the corners of the rounded rectangle.



## CSWire - Imported Cross Section

This template is used to modal a cross section imported from an external source. Wire cross sections are created automatically when [importing a shell](#) from a surface STL file. The imported shell consists of wire cross sections that describe the shape of the shell.

Each cross section that is imported from an external file source is saved as a wire cross section. The wire cross sections can then be viewed, modified, and saved using the cross section editor. For help using the cross section editor follow the link below.

[Cross Section Editor](#)





## CustomDimension - User Created Custom Dimension

This template is used to represent user defined dimensions, or custom dimensions. Dimensions can include length and angle dimensions in 3D or 2D views. Length and angle dimensions can be formed using control points and control lines.

### Main

---

#### Dimension Type

Specifies the type of dimension calculated. These include:

- **Length** indicates that the dimension is a length dimension.
- **Angle** indicates that the dimension is an angle dimension.





## DatumPlane - Datum Plane

This template is used to build reference (or datum) planes. Datum planes are defined as the plane perpendicular to a normal vector. The normal vector is given by 3 location attributes and 3 direction attributes. These datum planes can then be used as reference planes for moving components, dimensioning, building and dividing baffles, and discretization.

### Main

<b>Location X</b>	X location of the base of the normal vector. This will be the X location of the center of the datum plane.
<b>Location Y</b>	Y location of the base of the normal vector. This will be the Y location of the center of the datum plane.
<b>Location Z</b>	Z location of the base of the normal vector. This will be the Z location of the center of the datum plane.
<b>Direction X</b>	X component of the normal vector describing the datum plane.
<b>Direction Y</b>	Y component of the normal vector describing the datum plane.
<b>Direction Z</b>	Z component of the normal vector describing the datum plane.
<b>Use as Discretization Plane</b>	<p>Flag to use this datum plane as a discretization plane. If checked, then this plane will represent a firm boundary for the discretization routine. This will result in the model being discretized at the location of this plane.</p> <p>Since the discretization routine works along the 3 directions of the local shell's axis, a datum plane used as a discretization plane must be aligned in 1 of these 3 directions. If the datum plane is at an arbitrary orientation relative to the shell's local axis, then it cannot be used as a discretization plane and this attribute will be ignored.</p>
<b>Datum Plane Size</b>	Specifies the length and width to be used when drawing the datum plane in the graphical window.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the Default Transparency Percentage option in File → Options → General.
<b>Display Color</b>	<p>Indicates the color used when drawing the part. The color choices include:</p> <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li></ul>





- Grey
- Black







## **FESharedNodes - Node List for Imported Finite Element Mesh**

This template defines the local node positions for an imported 3D finite element mesh. This node list will be shared among any FE surface mesh or volume mesh that is contained within the imported mesh file. The surface or volume meshes that share the node list are displayed as children in the GEM3D model tree. All inputs are filled in automatically upon importing the mesh, therefore it should not be necessary to manually modify any of the attributes below.

### **Mesh Files and Nodes**

<b>Mesh File</b>	File name of the original imported FE mesh.
<b>Node Number</b>	A unique integer for the node. The numbering does not need to start from one, and does not need to be sequential. These values are typically created by GEM3D when the mesh is imported, are should not typically be modified manually.
<b>X-Coordinate</b>	X coordinate value of the specific node, in the local coordinate system for the mesh.
<b>Y-Coordinate</b>	Y coordinate value of the specific node, in the local coordinate system for the mesh.
<b>Z-Coordinate</b>	Z coordinate value of the specific node, in the local coordinate system for the mesh.





## **GEMAssyConn - Assembly Connection**

This template is used to represent an assembly connection. An assembly connection creates a connection between multiple components without moving them. This works like a group operation. When any one of the components of an assembly is moved, the other(s) will move along with it to maintain their relative location. This connection is typically used to connect a muffler shell with its internal pipes.





## GEMBaffle - Baffle

This template is used to model a baffle in a shell. Baffles separate shells into multiple chambers. A shell can contain multiple baffles in any particular location and orientation. Multiple baffles of the same type cannot be placed in the same location. Due to the complexity of validation across different baffle types and maintaining flexibility in model building it is the responsibility of the user to ensure multiple baffles of different types do not get placed in the same location as this will produce bad discretization results.

### Main

#### Baffle Thickness

Wall thickness of the baffle.

#### Baffle Type

Specifies the type of baffle to be used. Available types include:

- **Vertical** a vertical baffle with respect to the direction of extrusion of the shell (parallel to the shell cross-section). With this baffle type only the **Location along Shell** attribute below needs to be specified which will give the location of the vertical baffle along the extruded length of the shell.
- **Horiz-major** a horizontal baffle with respect to the direction of extrusion of the shell aligned with the major axis of the first cross section. With this baffle type only the **Location along Shell** attribute below needs to be specified. This will give the location of the baffle in the major direction, where the anchor point of the shell is 0 (the anchor point of the standard cross sections is the centroid).
- **Horiz-minor** a horizontal baffle with respect to the direction of extrusion of the shell aligned with the minor axis of the first cross section. With this baffle type only the **Location along Shell** attribute below needs to be specified. This will give the location of the baffle in the minor direction, where the anchor point of the shell is 0 (the anchor point of the standard cross sections is the centroid).
- **General** a general baffle that can be in any orientation inside the shell. With this baffle type the 6 location and direction attributes below must be specified. The 3 location attributes give the location of the baffle. The 3 direction attributes give the normal vector to the plane that will be the baffle.

#### Location along Shell

Location along the shell where the baffle will be placed. The **Baffle Type** specified above determines the orientation of this location. This location specifies the location of the front face of the baffle. The **Baffle Thickness** will then be extruded in the appropriate direction based on the **Baffle Type**. If **Baffle Type=General**, then this attribute must be set to "ign".

#### Location X in Shell

X location in the shell of the baffle. This specifies a relative location where the local coordinate system of the shell is 0.0. For normal shells ('GEMShell') this is the center of the first cross section. For mesh shells ('GEMMeshShell') this is given by the **Orientation** tab. This location specifies the location of the front face of the baffle. The **Baffle Thickness** will then be extruded in the appropriate direction based on the direction attributes below. If **Baffle Type** is NOT set to **General**,





then this attribute must be set to "ign".

#### Location Y in Shell

Y location in the shell of the baffle. This specifies a relative location where the [local coordinate system](#) of the shell is 0.0. For normal shells ('GEMShell') this is the center of the first cross section. For mesh shells ('GEMMeshShell') this is given by the **Orientation** tab. This location specifies the location of the front face of the baffle. The **Baffle Thickness** will then be extruded in the appropriate direction based on the direction attributes below. If **Baffle Type** is NOT set to **General**, then this attribute must be set to "ign".

#### Location Z in Shell

Z location in the shell of the baffle. This specifies a relative location where the [local coordinate system](#) of the shell is 0.0. For normal shells ('GEMShell') this is the center of the first cross section. For mesh shells ('GEMMeshShell') this is given by the **Orientation** tab. This location specifies the location of the front face of the baffle. The **Baffle Thickness** will then be extruded in the appropriate direction based on the direction attributes below. If **Baffle Type** is NOT set to **General**, then this attribute must be set to "ign".

#### Direction X in Shell

X component of the vector describing the normal to the baffle. Only a unit vector is needed to describe the normal, so the **Direction X in Shell**, **Direction Y in Shell**, and **Direction Z in Shell** attributes may be replaced with the equivalent unit vector. If **Baffle Type** is NOT set to **General**, then this attribute must be set to "ign".

#### Direction Y in Shell

Y component of the vector describing the normal to the baffle. Only a unit vector is needed to describe the normal, so the **Direction X in Shell**, **Direction Y in Shell**, and **Direction Z in Shell** attributes may be replaced with the equivalent unit vector. If **Baffle Type** is NOT set to **General**, then this attribute must be set to "ign".

#### Direction Z in Shell

Z component of the vector describing the normal to the baffle. Only a unit vector is needed to describe the normal, so the **Direction X in Shell**, **Direction Y in Shell**, and **Direction Z in Shell** attributes may be replaced with the equivalent unit vector. If **Baffle Type** is NOT set to **General**, then this attribute must be set to "ign".

## Visual

#### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

#### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green





- Copper
- Gold
- Grey
- Black

## Boundary

### Boundary Plane

Name of a [DatumPlane](#) object describing the plane to be used as the boundary. The datum plane specified here should intersect the baffle. The baffle will then be clipped according to the **Boundary Direction** specified below. The datum plane should be specified in the global coordinate system when being used as a boundary plane. If no boundary datum plane is desired, then this attribute may be set to "ign".

The most common use of this attribute is to specify another baffle as a boundary plane. The value selector (right-click menu) can be used to select the datum planes of any existing baffles. The datum plane's name will begin with the baffle name and end with dp#, where 1 is the front face and 2 is for the back face (since the baffle has wall thickness).

### Boundary Direction

Specifies which portion of the baffle will be kept relative to the specified boundary datum plane. The choices include:

- **Normal** indicates that the baffle will be kept on the same side that the normal vector to the boundary plane points.
- **Opposite** indicates that the baffle will be kept on the opposite side that the normal vector to the boundary plane points.





## GEMBaffleLeak - Leakage past a Baffle

This template is used to model the leakage past a baffle in a shell. The leakage is modeled as an orifice representing flow around the outside edge of the baffle along the surface of the shell. Depending on the construction style of the baffle, it is possible for gaps or leaks to occur around the edge. This is fairly common in mufflers and the baffles are not completely sealed around the outer edge and can therefore cause leakage between chambers. Although less common, this situation could exist for intake resonators that use molded separators between chambers for tighter packaging of multiple resonators. Leakage between chambers typically has a very significant effect on the individual shape of the response of the muffler.

### Main

☉ Leakage Area	Specifies the leakage area as a fraction of the total baffle area. This is a good method to use if the actual leakage gap is not known or not consistent. It can be changed to tune the model. Actual leakage areas will vary quite a bit due to exact geometry and construction, but typically range from 0 to 1 %.
☉ Leakage Gap	Specifies the leakage area as a gap around the baffle. This is a good method if the gap is known or can be found from drawings or CAD models. It can be change to tune the model. Actual leakage gaps will vary quite a bit due to exact geometry and construction, but typically range from 0 to 0.5 mm.

### Main

Forward Discharge Coefficient	Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.
Reverse Discharge Coefficient	Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Options

Forward End Correction (Length/Diameter)	End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does <u>not</u> recommend a
--	--





## Reverse End Correction (Length/Diameter)

user to enter another value under any circumstances. ("def" = 0.355)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

## Heat Conduction "Flange"

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

## Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one '[OrificeConn](#)' connection in a string of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def". The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery**

This option enables the pressure recovery in Orifice Connections as described in the Flow Manual, Chapter 2.2.8. If this option is used and discharge coefficients are specified, the resulting pressure drop may not be equal to Bernoulli's Law due to the pressure recovery calculated. **This is the default value which has been used by GT-Power in all previous versions.**





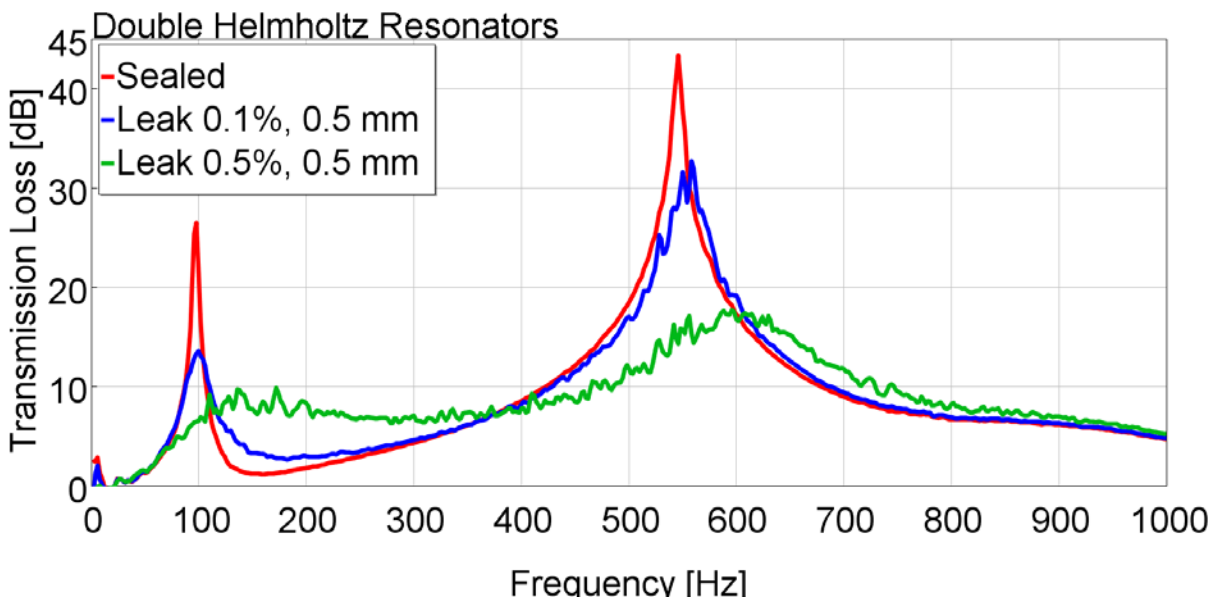
- **NoRecovery**

This option disables the pressure recovery in the Orifice Connection. With this choice, the pressure drop across the Orifice Connection is calculated according to Bernoulli's Law.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def".

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is not allowed.

**Notes about results:** Leakage between chambers in mufflers or resonators will tend to decrease the effective of the resonator and shift the design frequency. As an example of this, the plot below shows a single shell that contains 2 Helmholtz resonators that use a baffle to separate the 2 chambers that represent the volumes of the individual resonators. By design they should act at 100 Hz and 550 Hz, which they do very well when sealed (no leakage). As the leakage is increased, the effectiveness of them (peaks) tends to be reduced. As the leakage is increased further, the design frequency also starts to shift as the 2 resonators "share" some volume.







## GEMBPipe - Pipe with Bend

This template is used to model pipes that have a bend.

### Geometry

<b>Angle of Bend</b>	Angle of the bend measured from the start of the bend on each side. "def" may be entered if the pipe being represented consists entirely of a bend (no straight sections), and both the <b>Length of Bend</b> and <b>Radius of Bend</b> are specified. (0° indicates a straight pipe; 180° indicates a U-shaped bend.)
<b>Radius of Bend</b>	Radius of the pipe's bend measured from the center of curvature to the centroid of the pipe's cross-sectional area. "def" may be entered if the pipe being represented consists entirely of a bend (no straight sections), and both the <b>Length of Bend</b> and <b>Angle of Bend</b> are both specified.
<b>Bend Across</b>	<p>This attribute specifies the direction of the bend. The choices and their descriptions include:</p> <ul style="list-style-type: none"><li>• <b>Width (X)</b> indicates that the pipe will bend across the width of the specified cross section defining the pipe.</li><li>• <b>Height (Y)</b> indicates that the pipe will bend across the height of the specified cross section defining the pipe.</li></ul>
<b>Length</b>	Pipe length. "def" may be entered if the pipe being represented consists entirely of a bend (no straight sections), and both the <b>Radius of Bend</b> and the <b>Angle of Bend</b> are specified. (See note below about length and bends.)
<b>Wall Thickness</b>	Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a <a href="#">'WallThermalProperty'</a> reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the <a href="#">'WallThermalProperty'</a> reference object.

### Cross Sections

<b>◎ Cross Section Name</b>	Name of the cross section object describing the shape to be used for the bent pipe. The bent pipe allows any of the 4 standard cross section shapes ( <a href="#">circle</a> , <a href="#">ellipse</a> , <a href="#">rectangle</a> , <a href="#">rounded rectangle</a> ). A maximum of 2 cross sections may be specified. If only 1 is specified, then that cross section will be used at the beginning and end of the pipe. If 2 are specified, then the first one will be used at the beginning of the pipe and the second one will be used at the end. The resulting pipe will use a smooth transition to change cross sections along the length of the pipe.
<b>(◎) Diameter</b>	Pipe diameter.





## Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z</b>	Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.

## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

### Discretization Length

Discretization length to be used for pipes during the discretization process. This length does not need to be an even fraction of the entire





## Initial State Name

pipe length; the code will adjust to divide the pipe appropriately. A value of "def" will use the discretization length found in the global discretization window. See [export gtm](#) for additional information.

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

## Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

#### Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

### ☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

## Additional Geometry Options

Specify optional geometric characteristics of the flow component.

### Pipe Elevation Change or 3D Acceleration Object

This attribute defines either the elevation change of the pipe from the inlet to the outlet, or 3D acceleration resolved along the pipe axis via a 'PipeMotion3DBM' reference object. If "def" is entered, then the elevation change or 3D acceleration object of the pipe will be automatically generated from the option selected in the **Gravity/Acceleration Options** field of [File>Options>Discretization](#).





When the gravity option is used, a positive value means port 2 is at a higher elevation than port 1, and vice versa. The acceleration due to gravity is assumed to be  $9.80665 \text{ m/s}^2$ . When 'XYTable' reference object is used, X is defined as pipe length (which is normalized) and Y is the elevation change.

This attribute cannot be used in parallel with the **Body Force Acceleration** attribute under the Options folder.

## No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### ☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

### ☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

### ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This





option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses). This option should be used when this Pipe object will be an intake or exhaust port for a cylinder.

© **Adiabatic**

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

**Additional Thermal Options**

Specify optional thermal characteristics of the flow component.

**Heat Transfer Multiplier**

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

**Heat Input Rate**

The rate of heat input to the fluid or the name of a dependency reference object.

**Thermocouple Object**

Name of a ['Thermocouple'](#) reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

© **Heat Transfer Correlation (Colburn)**

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

© **User Defined Heat Transfer Model**

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to defined the name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

© **Heat Transfer Coefficient**

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

**Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)**

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat





is released entirely to the fluid.

- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.





### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

### User Defined Friction Model

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

## Acceleration Options

### Body Force Acceleration (along pipe axis)

Specify acceleration of the fluid in the flow component due to body forces such as gravity or centrifugal force.

## Pressure Loss Coefficients (Bend and Taper Losses)

### ☉ Determine Loss Coefficients (Fwd and Rev) from Geometry

Specify the influences of the bends and taper losses on the Pipe pressure drop. Please note that choosing any option other than **Determine Loss Coefficients (Fwd and Rev) from Geometry** will ignore the influence of user-specified bends and tapers in pressure drop calculations.

Calculates the pressure loss coefficient over the length of the pipe automatically based on the geometry (i.e. tapers and bends).

### ☉ Zero Pressure Losses from Bends and Tapers

Neglects all pressure drop effects due to bends and tapers. This is equivalent to specifying **Forward Loss Coefficient** and **Backward Loss Coefficient** as zero. Note that friction losses are computed separately.

### ☉ Forward Pressure Loss Coefficient

Pressure loss coefficient for flow from port 1 to port 2. This attribute is used to account for pressure losses due to geometry, such as bends and non-circular cross-sections. Enter "def" to have the pressure loss automatically calculated by the code using the cross-sectional shape and bend information that is entered. (0 indicates no additional pressure loss.)

### (☉) Reverse Pressure Loss Coefficient

Pressure loss coefficient in the opposite direction from port 2 to port 1 (see **Forward Pressure Loss Coefficient** above).

## Flexible Wall

If the checkbox below is activated, a 'PipeRoundFlexWall' object will be created to represent this part when discretization occurs.

### ☒ External Pressure

Pressure on the outside of the pipe walls ("def" = 1 bar). If the pipe diameter deformation is imposed with an 'XYTable' this attribute is ignored.







### (☒) Youngs Modulus

Young's modulus of elasticity for the pipe wall material or reference object. Allowed reference objects include 'RLTDependenceXY', 'RLTDependenceXYZ', 'ProfileTransient', and 'XYTable'. A single Young's modulus is applied to all subvolumes along the pipe unless an 'XYTable' is used. When an 'XYTable' is used to specify Young's Modulus, X is defined as the interior temperature of the innermost wall layer of the pipe (as opposed to the wall's internal surface temperature) in Kelvin. The unit of the Young's modulus Y variable in the 'XYTable' must be the same as the unit of the **Youngs Modulus** attribute. Note that for 'XYTable' input cases if the **Wall Temperature Solver Object** in the Main folder is set to "ign" (and therefore there is an imposed constant wall temperature) there will be no change in the applied Young's modulus in time.

If the pipe diameter deformation is imposed with an 'XYTable' this attribute is ignored.

### (☒) Poissons Ratio

Poisson's ratio of the pipe wall material. If the pipe diameter deformation is imposed with an 'XYTable' this attribute is ignored.

**Length and Bend Specifications:** If this component is being used to represent a bend with no straight sections, only two attributes are needed to describe the bend. Of the **Length**, **Radius of Bend**, and **Angle of Bend**, only two of these attributes need to be specified and the third may be set to "def". If all three attributes are specified, the **Length** must be equal to or greater than the length of the bend that will be calculated using the **Radius of Bend** and the **Angle of Bend**. If the user-specified length is greater than the calculated length of the bend, the difference will be modeled as a straight pipe and be assigned half to each side of the bend. If the user-specified length is less than the calculated length of the bend, an error message will be issued.







## GEMCap - End Cap

This template is used to add an end cap to an open port. This will block any flow from entering or exiting at the port. Caps can be placed on round pipes, straight pipes, bent pipes, T-splits, and Y-splits. Caps cannot be placed at ports that currently have an existing connection.

### Geometry

<b>Port ID</b>	Indicates the port number of the component where the cap will be placed. The choices include: <ul style="list-style-type: none"> <li>• 1</li> <li>• 2</li> <li>• 3 (<a href="#">'GEMTsplit'</a> and <a href="#">'GEMYsplit'</a> only)</li> </ul>
----------------	--

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>



## GEMCatalystBrick - Flow-Through Catalyst Model

This template is used to model a catalyst brick. The catalyst brick is treated as a bundle of passages, where each passage represents a cell in the catalyst brick. Chemical reactions are added later in GT-ISE. Please refer to the Aftertreatment Manual for more information about catalyst modeling. Catalyst components will be drawn with a triangle in the center of the cross section to differentiate it from other pipes in the graphical window.

### Geometry

#### Catalyst Brick

The following 3 attributes specify the details of the brick section of a catalyst component.

##### Cross Section Name

Name of the cross section object(s) describing the shape to be used for the catalyst shell. This cross section will determine the total frontal area of the catalyst brick. The size of the individual passages will be determined from this attribute and the **Cell Density** above. Any of the 4 standard cross section shapes (circle, ellipse, rectangle, rounded rectangle) are allowed. If only 1 cross section is specified, then that cross section will be used at the beginning and end of the pipe. If 2 or more are specified, then the first one will be used at the beginning of the pipe and the last one will be used at the end. The resulting pipe will use a smooth transition to change cross sections along the length of the pipe.

##### Length

Distance to the next cross section. The cross section specified above in the same column will be extruded for this distance and blended with the next specified cross section. Only a single cross section is necessary, in which case this value will be the catalyst length. When multiple cross sections are specified, this is the length of each section. Also with multiple cross sections, the value of the last distance will be ignored as there is not another cross section to extrude to.

##### Shell Wall Thickness for Visual Effects

Wall thickness of the catalyst shell for visual effects only. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in the Thermal folder. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the Thermal folder.

#### Entry and Exit Cones

The following 4 attributes specify the geometry of the entry and exit cones to the brick section of a catalyst component. When checked, cones will be included in the created component. This is the recommended method for including cones in a catalyst component.

##### ☒ Diameter at Inlet of Entry Cone

Inside diameter at the inlet of the entry cone. The diameter at the outlet of the entry cone will be equal to the equivalent diameter of the catalyst brick section, defined by the **Cross Section Name** attribute above.

##### ☒ Entry Cone Length

Length of the entry cone.

##### ☒ Diameter at Outlet of Exit Cone

Inside diameter at the outlet of the exit cone. The diameter at the inlet of the exit cone will be equal to the equivalent diameter of the catalyst brick section, defined by the **Cross Section Name** attribute above.





(☒) **Exit Cone Length** Length of the exit cone.

## Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z</b>	Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.

## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>

## Main

<b>Initial State Object</b>	Name of the 'FluidInitialState' reference object describing the initial conditions of the gas inside the catalyst.
-----------------------------	--





**☉ Standard Channel Geometry**

Use this radio button to model a flow-through catalyst with a standard channel shape.

**(☉) Channel Shape**

One of the following selections:

- **circle**
- **square**
- **triangle**
- **hexagon**

The **circle** selection is available for running models that were evolved from versions earlier than V2016, but it is recommended to use the **General Geometry** radio button for circular channels instead.

The **triangle** selection uses an equilateral triangle shape.

The **triangle** and **hexagon** selections may only be used for aftertreatment applications using the Advanced Adaptive chemistry solver. The **circle** selection is not allowed with the Advanced Adaptive chemistry solver.

**(☉) Cell Density**

Number of cells (channels) per frontal cross-sectional area.

**(☉) Substrate Wall Thickness**

Total wall thickness of the substrate material.

**(☉) Washcoat Layer 1 Thickness**

Average thickness of washcoat layer 1, which is always in contact with the channel gas. This layer thickness is included in the calculation of flow area and thermal mass. It may be set to "ign".

**(☉) Washcoat Layer 2 Thickness**

Average thickness of washcoat layer 2, which is located between the substrate and washcoat layer 1. This layer thickness is included in the calculation of flow area and thermal mass. This input is required for modeling a dual washcoat layer flow-through catalyst. It may be set to "ign".

**☉ General Geometry (Wrapped Metal, Packed Bed, etc.)**

Use this radio button to model a packed bed reactor, crushed monolith, or sinusoidal wrapped metal catalyst. When this radio button is selected the "Options" folder is completely hidden from view, because the attributes in the "Options" folder do not apply to the attributes below.

**(☉) Specific Area**

Internal surface area of the catalyst per total volume of the reactor (frontal area \* length).

**(☉) Solid Fraction of Substrate**

Solid fraction of the substrate material.

**(☉) Solid Fraction of Washcoat Layer 1**

Solid fraction of the washcoat layer 1 material, which is always in contact with the channel gas. This layer solid fraction is included in the calculation of flow area and thermal mass. It may be set to "ign".

**(☉) Solid Fraction of Washcoat Layer 2**

Solid fraction of the washcoat layer 2 material, which is located between the substrate and washcoat layer 1. This layer solid fraction is included in the calculation of flow area and thermal mass. This input is required for modeling a dual washcoat layer flow-through catalyst. It may be set



to "ign".

(☉) Friction Factor \*  
Reynolds Number

Friction Factor \* Reynolds Number

(☉) Nusselt Number

Nusselt Number at constant wall temperature, for determining the heat transfer coefficient.

An '[XYFunction](#)' object can be used to make the Nusselt Number a function of the Reynolds Number, local Re for AA solver, where X is the Reynolds Number and Y is the Nusselt Number. This function feature should only be used by advanced users that understand the ramifications of altering the Nusselt Number (we are not responsible for bad results generated by using this feature - use at your own risk).

(☉) Sherwood Number

Sherwood Number at constant wall temperature, for determining the diffusion mass transfer coefficient. Set to "def" to make it equal to the Nusselt Number above.

An '[XYFunction](#)' object can be used to make the Sherwood Number a function of the Reynolds Number, local Re for AA solver, where X is the Reynolds Number and Y is the Sherwood Number. This function feature should only be used by advanced users that understand the ramifications of altering the Sherwood Number (we are not responsible for bad results generated by using this feature - use at your own risk).

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow-through catalyst. In most cases, the **Calculated Wall Temperature** option will give the most realistic results.

(☉) Imposed Wall  
Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and the fluid will be calculated using this imposed wall temperature, along with the actual local fluid temperature and heat transfer coefficient in each subvolume.

(☉) Calculated Wall  
Temperature

Solve for the substrate and outer layer wall temperatures, and the resulting heat transfer rate to the fluid, accounting for wall material properties and external boundary conditions surrounding the catalyst.

(☉) Initial Wall  
Temperature

Temperature of the substrate and outer layer walls at the first time step of the simulation.

(☉) Substrate Thermal  
Properties Object

Name of the '[MaterialThermalProp](#)' reference object defining the material properties of the substrate for axial conductivity, density, and specific heat. The conductivity and density should include the porosity of the substrate, meaning (1-porosity)\*(bulk solid property). Predefined reference objects for substrate materials Cordierite and SiliconCarbide are available in the template library.

(☉) Washcoat Layer 1  
Thermal Properties

Name of the '[MaterialThermalProp](#)' reference object defining the material properties of washcoat layer 1 for axial conductivity, density,



<b>Object</b>	and specific heat. The conductivity and density should include the porosity of the layer, meaning $(1-\text{porosity}) \times (\text{bulk solid property})$ . A predefined reference object for the common washcoat material Alumina is available in the template library. This attribute may be set to "ign" if the corresponding washcoat layer thickness or solid fraction is "ign".
<b>(◎) Washcoat Layer 2 Thermal Properties Object</b>	Name of the ' <a href="#">MaterialThermalProp</a> ' reference object defining the material properties of washcoat layer 2 for axial conductivity, density, and specific heat. The conductivity and density should include the porosity of the layer, meaning $(1-\text{porosity}) \times (\text{bulk solid property})$ . A predefined reference object for the common washcoat material Alumina is available in the template library. This attribute may be set to "ign" if the corresponding washcoat layer thickness or solid fraction is "ign".
<b>(◎) Outer Wall Layers Thermal Properties Object</b>	<p>Name of a '<a href="#">WallThermalProperty</a>' object defining the emissivity, thickness, and material thermal properties of each outer layer. For example metal canning, insulation, air gap, heat shield, etc. Outer Layer 1 is in direct contact with the cylindrical surface of the substrate. This may be set to "ign" to ignore the thermal mass and resistance to external heat transfer of outer layers.</p> <p>If outer layer 1 is using a '<a href="#">MaterialThermalProp</a>' object with <b>Opacity</b> set to transparent, the radiation heat transfer for layer 1 will be ignored.</p> <p>Direct convective heat loss from the substrate to the ambient can be modeled by setting this attribute to "ign" and defining a '<a href="#">WallThermalBoundary</a>' object for the attribute <b>External Boundary Conditions Object</b> below with the <b>External Convection Coefficient</b> or <b>Free Convection Model</b> attributes defined in the object.</p>
<b>(◎) External Boundary Conditions Object</b>	<p>Name of a '<a href="#">WallThermalBoundary</a>' object defining the external boundary conditions for calculating heat transfer to the ambient by convection and/or heat transfer to a black body by radiation.</p> <p>Direct convective heat loss from the substrate to the ambient can be modeled by defining this object, and setting the <b>Outer Wall Layers Thermal Properties Object</b> to "ign" in the attribute above.</p>
<b>◎ Wall Temperature from Connected Thermal Primitive</b>	Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection (' <a href="#">ConvectionConn</a> ') to a thermal primitive part (i.e. ' <a href="#">ThermalMass</a> '). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).
<b>Additional Thermal Options</b>	Specify optional thermal characteristics of the flow-through catalyst.
<b>Heat Transfer Multiplier</b>	Heat transfer coefficient multiplier. The calculated heat transfer rate between the internal fluid and the wall will be scaled by this factor. . ("def"=1.0) (Also see the Time Step and Solution Control Object in the FlowControl folder of Run Setup for information about the global "Heat





Transfer Enhancement Factor")

If the **Global Heat Transfer Multiplier: constant term** in the [FlowControlExplicit](#) object is set to 2.625, then the local Heat Transfer Multiplier here should be set to  $1/2.625=0.381$  to compensate.

This attribute is hidden when the **General Geometry** radio button in the "Main" folder is selected since the **Nusselt Number** will be specified directly.

## Heat Input Rate

The rate of heat input to the gas or substrate based on the **Heat Input Location** attribute below, or the name of a dependency reference object. For most catalyst applications, this attribute should not be needed and should be set to "ign".

This attribute combined with the two below may be used to model an electrically heated catalyst. This attribute may also be used for calibrating 1D heat loss, by making the heat input rate negative and location "substrate" in order to remove heat directly from the substrate. When this attribute is defined, and **Heat Input Location** is set to "substrate" then the **Calculated Wall Temperature** must be selected. The Heat Input Rate may also be actuated via an ActuatorConn for use in a control system, and the actuated value will overwrite the attribute value.

## Heat Input Axial Distribution

The normalized axial distribution of the heat input rate. The value "ign" means a uniform distribution. An [XYTable](#) may be used to specifying the normalized axial distribution of the heat input rate. The total heat input rate will be conserved such that the integral of the axial distribution will be equal to the total heat input rate value above.

This attribute must be set to "ign" when using Advanced Adaptive chemistry solver.

## Heat Input Location

The location that the heat input rate will be applied:

- **gas**
- **substrate**

This attribute must be set to **substrate** when using Advanced Adaptive chemistry solver.

## Heat Transfer User Model

The name of the [UserCodeReference](#) object which will be used to calculate the internal Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value above.

This attribute is hidden when the **General Geometry** radio button in the "Main" folder is selected since the **Nusselt Number** will be specified directly.







## Modeling Options

### 2D/3D Flow Discretization Method

One of the following choices pertaining to 2D/3D modeling only:

- **component(QS)** indicates that a single catalyst will be used for the solution. This must be selected when flow solver is set to **quasi-steady**.
- **compound(non-QS)** indicates that an internal compound of multiple catalysts will be created. This must be selected when flow solver is set to **explicit** or **implicit**.

### 2D/3D Model Object

Name of the 'ExhaustATDevice2D' or 'ExhaustATDevice3D' reference object defining the 2D or 3D catalyst model. This attribute may be set to "ign".

This attribute must be set to "ign" when using Advanced Adaptive chemistry solver.

### 1D Plot Frequency (seconds)

The frequency, in seconds, in which data will be stored and reported for the plots listed as '1D' located in the 'CatalystBrick' part. If this value exceeds the simulation duration no 1D plots will be stored. ("ign" = no 1D plots will be stored)

## Options

The attributes in this folder are optional modifiers, which only apply when the **Standard Channel Geometry** is selected in the "Main" folder. When the **General Geometry** option is selected in the "Main" folder, this "Options" folder is completely hidden from view, because these attributes do not apply.

### Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. ("def"=1.0)

Typically for square channel catalysts no adjustment is necessary for the linear pressure drop term. OrificeConn CD's can be adjusted for quadratic effects.

If the **Global Friction Multiplier: constant term** in the 'FlowControlExplicit' object is set to 1.875, then the local Friction Multiplier here should be set to  $1/1.875=0.533$  to compensate.

### Sherwood Number Override

Often referred to as the dimensionless mass transfer Nusselt number, this value represents the ratio of the convective mass transfer coefficient to the diffusion mass transfer coefficient. This value will effectively control the convection mass transfer coefficient, and is only used if the attribute **Diffusion** in the attached \*Reactions object is set to **quasi-steady**. The Sherwood number often correlates directly to the Nusselt number, meaning both have the same dimensionless number. The value of the Sherwood Number and Nusselt Number used by the solver depends on the Channel Shape attribute.





For **circle**, Nusselt number and "def" Sherwood number are both 3.66.

For **square**, Nusselt number and "def" Sherwood number are both 3.0.

There is no Nusselt number attribute for Standard Channel Geometry, so for other channel geometries, if you change the Sherwood Number you should also change the Heat Transfer Multiplier accordingly.

An '[XYFunction](#)' object can be used to make the Sherwood Number a function of the Reynolds Number, local Re for AA solver, and average Re for BDF solver, where X is the Reynolds Number and Y is the Sherwood Number. This function feature should only be used by advanced users that understand the ramifications of altering the Sherwood Number (we are not responsible for bad results generated by using this feature - use at your own risk). When using this function feature, the Nusselt Number ignores the function and continues to use the constant value for the given channel geometry selection of circle or square, breaking the link with Sherwood Number.

## Surface Reaction Area Ratio

Total reaction surface area divided by the nominal wall surface area. The nominal surface area is determined from the channel dimensions and channel flag assuming the surface is smooth. The channel dimensions are determined from the attributes above and are reported as RLT variables in the "Init" folder (diameter, length, and number of pipes).

The value of this attribute is simply an overall multiplier to the reaction rate. This attribute is primarily used to account for the increase in surface area, relative to a smooth surface, which results from the porous nature of the walls and washcoat, but it can also be used to account for additional surface area for non-circular channels. This attribute may be set to "ign" when no catalytic surface reactions are being modeled. ("def"=12.5, comes from reference in '[Cat3WayModel](#)', "ign"=1)

This attribute value is only used in the following situations: if **Catalyst Model Object** is defined as a '[Cat3WayModel](#)', if there is an attached '[SurfaceReactions](#)' with **Rate Expression Basis** set to "area", or if there is an attached '[GlobalReactions](#)' with **Reaction Site** set to "surface" and **Rate Expression Basis** set to "area". It is not used if **Reaction Site** is set to "gas" or if **Rate Expression Basis** is set to "volume" or "site (turnover number)". It is also not used for **Catalyst Model Object** defined with a '[CatOxModel](#)' reference object, because surface area is not an input to that model.

## Forward Pressure Loss Coefficient (Obsolete)

Pressure loss coefficient for flow from port 1 to port 2. This attribute is historically used to account for pressure losses due to geometry changes like bends and tapers (0 indicates no additional pressure loss.) "def" may be entered to let the code calculate losses due to bends and tapers, but this typically does not apply to catalysts. However, this attribute can be used to increase the quadratic pressure drop term associated with contraction and expansion pressure loss at the front and rear of the cat, respectively. Typical values for sum of contraction and expansion pressure loss coefficients are 0.4-0.5 for CatalystBrick.





	<p>This attribute must be set to "def" when using Advanced Adaptive chemistry solver.</p>
<b>Reverse Pressure Loss Coefficient (Obsolete)</b>	<p><b>This attribute is obsolete in the context of a catalyst and should not be used. Leave set to 'def'.</b> Pressure loss coefficient in the opposite direction from port 2 to port 1 (see <b>Forward Pressure Loss Coefficient</b> above).</p> <p>This attribute must be set to "def" when using Advanced Adaptive chemistry solver.</p>
<b>Friction User Model</b>	<p>The name of the <a href="#">'UserCodeFReference'</a> object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the <b>Friction Multiplier</b> value.</p>
<b>Catalyst User Model</b>	<p>The name of the <a href="#">'UserCodeFReference'</a> object for the user-sub-routine model code that creates a completely user-defined catalyst model including chemical kinetics. If a user model is not going to be used this attribute should be set to "ign".</p> <p>If evolving an older model, the reference objects <a href="#">'Cat3WayModel'</a> and <a href="#">'CatOxModel'</a> may exist in this attribute and solver will run, however the user cannot create new objects from these two templates. These reference objects are very simple and limited, and therefore <b>NOT</b> recommended for performing detailed accurate catalyst modeling. For detailed accurate catalyst modeling use either <a href="#">'SurfaceReactions'</a> or <a href="#">'GlobalReactions'</a>.</p>
<b>Material for Default Surface Roughness (Obsolete)</b>	<p><b>This attribute is obsolete in the context of a catalyst and should not be used. Leave set to user_value.</b> This attribute gives choices of materials that may be used to specify the surface roughness. However, since flow in a catalyst is by design laminar the friction (friction factor) and heat transfer (Nusselt number) are independent of the surface roughness. The choice of this attribute has no effect when used properly (Channel Re&lt;2000) and can be ignored.</p>
<b>Surface Roughness (Obsolete)</b>	<p><b>This attribute is obsolete in the context of a catalyst and should not be used. Leave set to 'def'.</b> Surface roughness attribute using the "Sand Roughness" height. (Sand Roughness height divided by the diameter of the pipe is "Relative Roughness" e/D used in the Moody Diagram).</p>





## **GEMChamber - Chamber of a Shell**

This template is used to represent a chamber of a shell. This template cannot be modified as its properties are a direct function of the parent shell.

---

### **Visual**

---

#### **Transparency Percent**

Indicates the transparency level used when drawing the part. 0 indicates opaque (solid) and 90 indicates almost completely transparent. For the chamber this value cannot be edited and the value will be based on the value from the parent component.

#### **Display Color**

Indicates the color used when drawing the part. For the chamber this value cannot be edited and the value will be based on the value from the parent component. The color choices include:

- **Red**
- **Blue**
- **Dark Blue**
- **Green**
- **Copper**
- **Gold**
- **Grey**
- **Black**





## **GEMConnection - Connection**

This template is used to represent the connection between 2 components. The attached components will have a solid connection so that the components will move together and be connected upon discretization. This connection can only be created by completing a connection operation ([flow connection](#), [extruded connection](#), [assembly connection](#), or [component flow connection](#) and cannot be modified.

### **Main**

### **Visual**

---

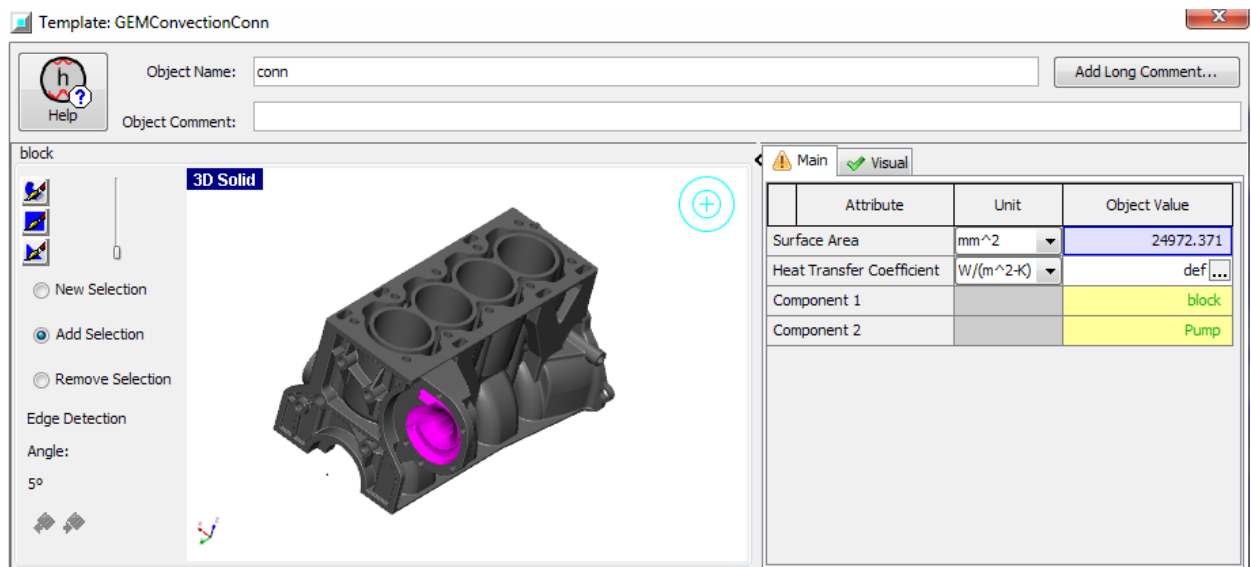
<b>Transparency Percent</b>	Indicates the transparency level used when drawing the part. 0 indicates opaque (solid) and 90 indicates almost completely transparent.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"><li>• <b>Red</b></li><li>• <b>Blue</b></li><li>• <b>Dark Blue</b></li><li>• <b>Green</b></li><li>• <b>Copper</b></li><li>• <b>Gold</b></li><li>• <b>Grey</b></li><li>• <b>Black</b></li></ul>



## GEMConvectionConn – Convection between Thermal Mass and Volume

This template enables convection heat transfer between a thermal mass and a fluid volume, and can be created from either a thermal mass or flow template by selecting the **Add Convection Connection** option and then clicking on the corresponding component(s) that should be connected.

Note that if multiple components are selected, multiple connections will be made in the tree but no windows will appear for editing individual connections (they can be opened individually from the tree). If only a single connection is requested, a window showing the connection will appear similar to below. This window allows the intersecting surfaces between the mass and the volume to be edited as needed.



The options on the left-hand side control whether a mouse-click on the object will add or remove triangles from the selection.

The slider on the left controls the “Edge Detection Angle”. The minimum value of 0 degrees will result in a mouse-click painting only the triangle that was clicked on. As the angle increases, the painting will extend to adjacent triangles until the angle of the edge between triangles exceeds the threshold.

### Main

Surface Area

The surface area of the painted selection. Although this can be directly entered in this field, this value should generally be determined automatically by GEM3D based on the selection painted in the 3D window. The value will automatically update as surface elements are added or removed from the selection.

Heat Transfer Coefficient

Convection heat transfer coefficient between the mass and the volume. This value will be passed to any ConvectionConn part that is created in the exported model as a result of this connection. A value of “def” implies that the heat transfer coefficient will come from the adjacent flow volume (pipe or flowsplit).





**Component 1** Not editable. The first component to be connected is listed for information.

**Component 2** Not editable. The second component to be connected is listed for information.

---

## Visual

---

**Transparency Percent** Indicates the transparency level used when drawing the part. 0 indicates opaque (solid) and 90 indicates almost completely transparent.

**Display Color** Indicates the color used when drawing the part. The color choices include:

- **Red**
- **Blue**
- **Dark Blue**
- **Green**
- **Copper**
- **Gold**
- **Grey**
- **Black**

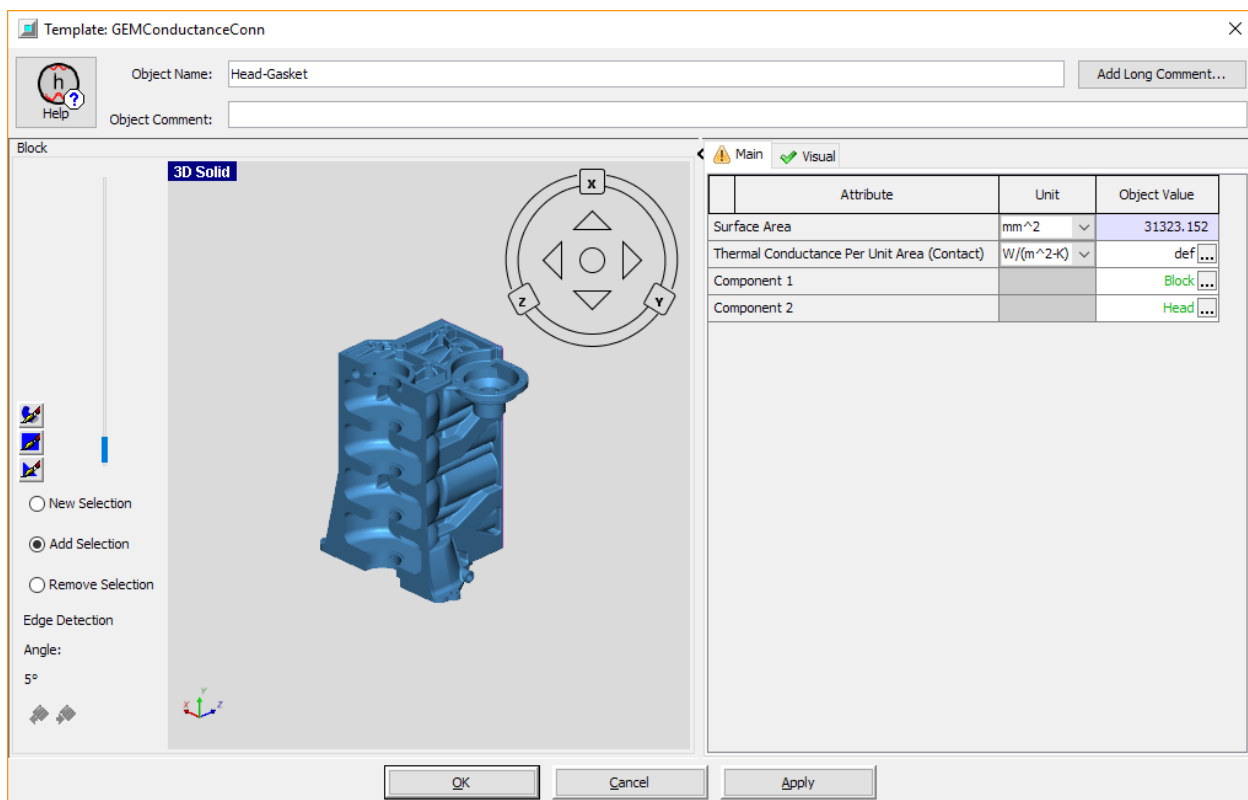




## GEMConductanceConn – Conduction between Thermal Mass(es) and EngCylStrucCond

This template enables convection heat transfer between a thermal mass and another thermal mass or EngCylStrucCond, and can be created from either a thermal mass or EngCylStrucCond template by selecting the **Add Conduction Connection** option and then clicking on the corresponding component(s) that should be connected.

Note that if multiple components are selected, multiple connections will be made in the tree but no windows will appear for editing individual connections (they can be opened individually from the tree). If only a single connection is requested, a window showing the connection will appear similar to below. This window allows the intersecting surfaces between the masses to be edited as needed.



The options on the left-hand side control whether a mouse-click on the object will add or remove triangles from the selection.

The slider on the left controls the “Edge Detection Angle”. The minimum value of 0 degrees will result in a mouse-click painting only the triangle that was clicked on. As the angle increases, the painting will extend to adjacent triangles until the angle of the edge between triangles exceeds the threshold.

### Main

#### Surface Area

The surface area of the painted selection. Although this can be directly entered in this field, this value should generally be determined





automatically by GEM3D based on the selection painted in the 3D window. The value will automatically update as surface elements are added or removed from the selection.

**Thermal Conductance  
Per Unit Area (Contact)**

Conduction heat transfer coefficient between the mass and the volume. This value will be passed to any ConductanceConn part that is created in the exported model as a result of this connection. A value of “def” implies that there will be a large conductance value (1e9) will be used.

**Component 1**

Not editable. The first component to be connected is listed for information.

**Component 2**

Not editable. The second component to be connected is listed for information.

---

**Visual**

---

**Transparency Percent**

Indicates the transparency level used when drawing the part. 0 indicates opaque (solid) and 90 indicates almost completely transparent.

**Display Color**

Indicates the color used when drawing the part. The color choices include:

- **Red**
- **Blue**
- **Dark Blue**
- **Green**
- **Copper**
- **Gold**
- **Grey**
- **Black**







## GEMCrankshaft – Crankshaft Builder

This template is used generate a cranktrain model (.gtm file) from an existing CAD file of a crankshaft. In order to use this template, a CAD file must be supplied for the crankshaft and it has to be in solid format (not a surface format like STL). With this template, GEM3D will be able to generate a detailed cranktrain model that contains componets, such as Journal, Crank-Web, Crank-Pin, Shaft-Segment, Split-Pin, and Flywheel with most of the attribute values automatically calculated, and all parts automatically connected.

Typically, the crankshaft CAD consists of one single solid shape. In order to use this template, the user needs to slice the crankshaft solid shape into multiple shapes representing different components of a crankshaft. While slicing the crankshaft shape into different components, it is recommended to name the shapes to something appropriate so that it will be easier to identify the proper shape in the GEMCrankshaft template.

A tutorial is available to guide a user through generating a cranktrin model from a CAD. It can be accessed from the Help Menu → Tutorials → Graphical\_Applications → GEM3D → GEM3D-tutorials.pdf.

### Main

<b>Crank Rotation Direction from Front</b>	<p>The direction in which the crank rotates when looking from the front of the crank:</p> <ul style="list-style-type: none"> <li>• <b>Counter-Clockwise</b></li> <li>• <b>Clockwise</b></li> </ul>
<b>Shape Name Representing Crank Origin</b>	<p>The name of the solid shape representing the origin of the crank.</p>
<b>Crank Material Properties Object</b>	<p>The name of 'MaterialMechanical' object describing the material properties of the crank. The density property from the selected 'MaterialMechanical' object will be used to calculate the masses for the components, such as 'CrankPin', 'CrankWeb', 'Flywheel', 'Journal', 'ShaftSegment', and 'SplitPinTransPlate' components</p> <p>Standard materials area available in the template library, and they can be accessed by right-clicking and selecting "Value Selector".</p>

### Components

<b>Type (front to back)</b>	<p>The component type representing each shape. Note that the order needs has to be from the front of the crank to the back. The options are:</p> <ul style="list-style-type: none"> <li>• <b>Journal</b> should be used when the shape needs to be converted into a 'Journal' object.</li> <li>• <b>CrankWeb</b> should be used when the shape needs to be converted into a 'CrankWeb' object.</li> <li>• <b>CrankPin</b> should be used when the shape needs to be converted into a 'CrankPin' object.</li> </ul>
-----------------------------	--





- **Flywheel** should be used when the shape needs to be converted into a 'Flywheel' object.
- **ShaftSegment** should be used when the shape needs to be converted into a 'ShaftSegment' object.
- **SplitPinTransPlate** should be used when the shape needs to be converted into a 'SplitPinTransPlate' object.

#### Shape Name

The name of the solid shape representing the component type selected in the **Type (front to back)** attribute.

#### Cylinder Number

The cylinder number attached to the crankpin component. The cylinder number should only be defined when the **Type (front to back)** attribute is set to CrankPin and for any other components, it should be set to "ign". Please see figure 1 for an example of how to define this folder.

### Cylinders

#### Cylinder Number Firing Order

Cylinder numbers in the order in which they fire.

#### Firing Intervals Between Cylinders (First Row=0)

Engine firing angle relative to the preceding cylinder. For the first cylinder (not necessarily cylinder 1\_, this number must to be set to zero.

#### Cylinder Axis Angle (Viewed from Front, CCW, From Horizontal)

Cylinder axis angle with respect to the horizontal axis when viewed from the front of the crank. ...

Additional Notes



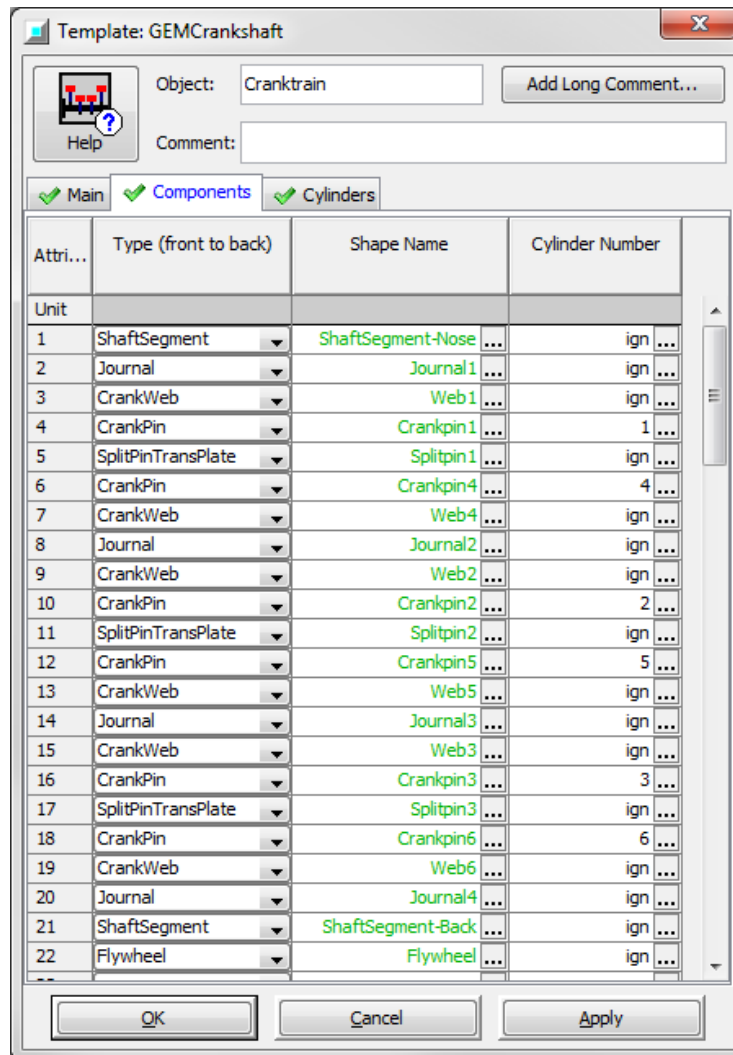


Figure 1 : Example of how to define the 'Components' folder





## **GEMFESharedNodes – Imported Finite Element Node List**

This template defines a list of nodes that are imported from an external finite element mesh file. The surface meshes or volume meshes that were imported as part of the same external FE mesh file will reference the same FESharedNodes object. Although all attributes are documented below, they are filled in automatically by GEM3D upon importing the FE mesh file, and generally should not be edited manually.

### **Mesh File and Nodes**

<b>Mesh File</b>	Name of the ' <a href="#">FESharedNodes</a> ' object that contains the node list (X, Y, Z coordinate locations of the nodes used by the surface elements). Note that this object is displayed in this field like a reference object, but it is displayed in the GEM3D model tree as a “parent” component above the surface mesh.
<b>Node Number</b>	The number assigned to the imported node. This number should match the one from the source finite element file.
<b>X-Coordinate</b>	The x coordinate of the specified node. This attribute should be automatically filled in by the import process.
<b>Y-Coordinate</b>	The y coordinate of the specified node. This attribute should be automatically filled in by the import process.
<b>Z-Coordinate</b>	The z coordinate of the specified node. This attribute should be automatically filled in by the import process.





## GEMFESurfaceMesh – Imported Finite Element Surface Mesh

This template defines a finite element surface mesh, imported from an external FE mesh file. This component may share a node list '[FESharedNodes](#)' with other surface meshes or volume meshes that were imported as part of the same external FE mesh file. Although all attributes are documented below, they are filled in automatically by GEM3D upon importing the FE mesh file, and generally should not be edited manually, with the exception of the Visual folder.

### Nodes

<b>Nodes</b>	Name of the ' <a href="#">FESharedNodes</a> ' object that contains the node list (X, Y, Z coordinate locations of the nodes used by the surface elements). Note that this object is displayed in this field like a reference object, but it is displayed in the GEM3D model tree as a “parent” component above the surface mesh.
--------------	--

### Elements

<b>Element No.</b>	Unique integer for each surface element. The numbering does not need to start from one, and does not need to be sequential.
<b>Element Property ID</b>	Integer value that identifies the physical property set for the element. During the GEM3D import the first row will be set to “def” which means that all elements are assumed to have the same properties (common). If this is not the case, values can be entered manually either in this template, or in the resulting objects in the exported model.
<b>Node 1-8</b>	The set of node numbers from the shared nodes list defining each element. Unused columns should be filled with “ign”. If a complete column is not needed, then only the first row may be set to “ign”.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> in File → Options.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>





## GEMFEVolumeMesh – Imported Finite Element Volume Mesh

This template defines a finite element volume mesh, imported from an external FE mesh file. This component may share a node list '[FESharedNodes](#)' with other surface meshes or volume meshes that were imported as part of the same external FE mesh file. Although all attributes are documented below, they are filled in automatically by GEM3D upon importing the FE mesh file, and generally should not be edited manually, with the exception of the Visual folder.

The [GEMFEVolumeMesh](#) component can be converted to a GEM Mechanical Flexible Body (Finite Element) ([GEMMechSolid3D](#)) or a GEM Thermal Finite Element Body ([GEMThermalFE](#)) using the standard 'Convert Shape to Component' option.

### Nodes

<b>Nodes</b>	Name of the ' <a href="#">FESharedNodes</a> ' object that contains the node list (X, Y, Z coordinate locations of the nodes used by the volume elements). Note that this object is displayed in this field like a reference object, but it is displayed in the GEM3D model tree as a “parent” component above the volume mesh.
--------------	--

### Elements

<b>Element No.</b>	Unique integer for each volume element. The numbering does not need to start from one, and does not need to be sequential.
<b>Element Property ID</b>	Integer value that identifies the physical property set for the element. During the GEM3D import the first row will be set to “def” which means that all elements are assumed to have the same properties (common). If this is not the case, values can be entered manually either in this template, or in the resulting objects in the exported model.
<b>Node 1-8</b>	The set of node numbers from the shared nodes list defining each element. Unused columns should be filled with “ign”. If a complete column is not needed, then only the first row may be set to “ign”.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> </ul>





- Gold
- Grey
- Black





## GEMFlowDirection - Flow Port Direction

This template is used to specify the details for a flow port direction. This template should be used on a boundary flow component that does not contain an external subassembly connection. The port number for connections and the flow direction can be specified here.

### Main

<b>Port ID</b>	Specifies the port ID number to be used to define the flow direction. This value can be any positive integer within the number of ports on the flow component.
<b>Flow Direction</b>	Specifies the desired connection direction to be used with the flow component. This direction will be used by the discretization routine to determine the connection direction in the rest of the model. The choices include: <ul style="list-style-type: none"> <li>• <b>Inlet</b> Specifies that the connection will point from the subassembly connection into the flow component. This should be chosen for an inlet boundary condition.</li> <li>• <b>Outlet</b> Specifies that the connection will point from the flow component into the subassembly connection. This should be chosen for an outlet boundary condition.</li> </ul>

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• <b>Red</b></li> <li>• <b>Blue</b></li> <li>• <b>Dark Blue</b></li> <li>• <b>Green</b></li> <li>• <b>Copper</b></li> <li>• <b>Gold</b></li> <li>• <b>Grey</b></li> <li>• <b>Black</b></li> </ul>

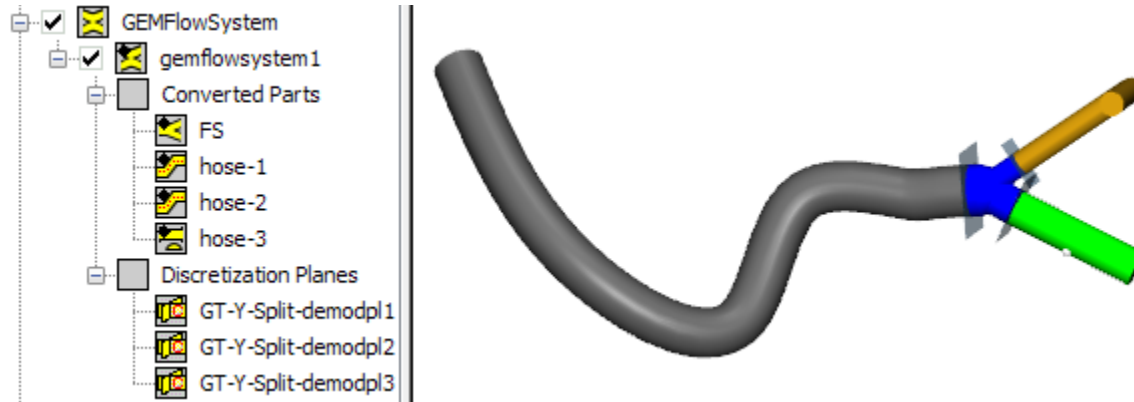




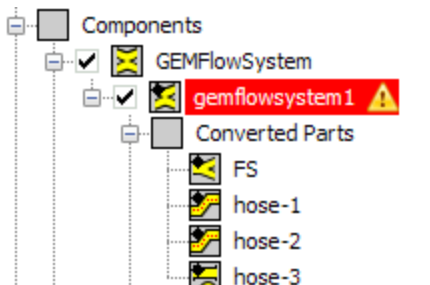


## GEMFlowSystem – System of Multiple Pipes and Flowsplits

This template represents a system of multiple pipes and flowsplits, converted from imported 3D geometry, with the divisions between individual pipes and flowsplits of the system defined by user positioned discretization planes. The original imported geometry data is stored with the flow system, along with the discretization planes.



An important benefit of this approach is that the effort required to convert a system can be largely re-used in the future. For example, if it becomes necessary to move a discretization plane, the plane can simply be moved in the 3D window. After moving the plane, the flow system will be flagged in the model tree as shown below to indicate that re-conversion is required. Re-opening the flow system and clicking “Finish” will reconvert the system to account for the moved discretization plane.



When a system is re-converted, any part specific attribute values that were previously entered (part names, overrides to converted geometry etc.) will be over-written ONLY if that part of the system needed reconversion as a result of the changes to the discretization planes.

The planes may also be copied to/from other flow systems or any other part that supports child datum planes. This means that if the CAD geometry is updated for the system, all of the discretization planes can be easily copied to the newly imported shape.

The attributes below are all stored at the top “Flow System” level of the tree, and represent the shared, non-geometric attributes of the system. These attributes will be applied to ALL exported pipes and flowsplits. Note that the geometric attributes are stored at the lower “Converted Parts” level of the tree. These attributes are specific to each template, and are not documented here.





## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>
<b>Display Child Discretization Planes</b>	<p>If “on”, all discretization planes stored for the flow system will be displayed in the 3D canvas. Otherwise they will be visually suppressed. The planes are always used for discretizing the flow system, whether displayed or not. Planes must be deleted if they should not be used for discretization.</p> <p>This option does not affect how the system is displayed within the flow system dialogs, where the planes will always be displayed.</p>

## Main

### Basic Geometry and Initial Conditions

<b>Discretization Length (for Pipes)</b>	Discretization length to be used for all pipes within the flow system during the discretization process. A value of “def” will use the discretization length found in the global discretization window. See <a href="#">export gtm</a> for additional information.
<b>Initial State Name</b>	Name of the 'FluidInitialState' reference object describing the initial conditions inside all volumes of the flow system.
<b>Surface Finish</b>	Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but also has an influence on the heat transfer coefficient, and therefore the fluid temperature.
<input checked="" type="radio"/> Smooth	Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).
<input type="radio"/> Roughness from Material	This attribute gives choices of materials that may be used to specify the surface roughness.





#### Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

### ☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram).

A numeric value or a parameter may be entered for this attribute.

#### Additional Geometry Options

### Pipe Elevation Change or 3D Acceleration Object

In the converted pipe parts of the flow system, this attribute will define either the elevation change of the pipe from the inlet to the outlet, or 3D acceleration resolved along the pipe axis via a '[PipeMotion3DBM](#)' reference object.

Because this attribute value is shared for all pipes in the flow system, this input should generally be set either to "def" or "ign"

If "**def**" is entered, then the elevation change or 3D acceleration object will be automatically generated for each pipe in the flow system from the option selected in the **Gravity/Acceleration Options** field of [File>Options>Discretization](#).

If "**ign**" is entered, then the elevation change or 3D acceleration will not be considered for any part in the flow system.

## Thermal

#### Wall Temperature Method

### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer





## ☉ Calculated Wall Temperature

coefficient in each subvolume.

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

## ☉ Wall Layer Properties Object

Name of the '[WallThermalProperty](#)' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

## ☉ Wall External Boundary Conditions Object

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

## ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

## ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

## ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth wall will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

### Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

### Thermocouple Object

Name of a '[Thermocouple](#)' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.





☉ Heat Transfer  
Correlation (Colburn)

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

☉ User Defined Heat  
Transfer Model

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the 'UserModel' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

☉ Heat Transfer  
Coefficient

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

Condense/Evaporate  
Water Vapor (Non-  
Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to 'FluidRefrigerant' Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an 'EjectorConn' object if





desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity 'SensorConn' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

### Flowsplit HTC Calculation Method

#### Method #1 (v7.5 and prior)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, and then they are weighted by the boundary area. This method tends to under predict the heat transfer coefficient in the flowsplit, and is only recommended to reproduce previous version results.

#### Method #2 (Recommended)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, which can be based on the expansion diameter or a custom dimension. An enhancement factor based on the flowsplit characteristic length is then added to each port value, and then the resulting coefficients are averaged based on the boundary flow rates. This method produces a more accurate prediction for the heat transfer coefficient, and is recommended especially in applications where the heat transfer in the flowsplit is a significant part of the overall circuit heat transfer rate.

#### Characteristic Dimension for Port HTC

This attribute will only be used when Method #2 is selected. The attribute selects which value will be used as the diameter in the heat transfer coefficient calculation. This can either be the Expansion Diameter, which will also be used for the pressure loss calculation, or the User Dimension which can be defined in the Boundary Data for each port. The User Dimensions will only affect the heat transfer coefficient calculation; it will not be used in the pressure loss calculation.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.





☉ **Friction Multiplier**

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

☉ **No Friction Pressure Losses**

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

**User Defined Friction Model**

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

**Acceleration Options (for Pipes)**

Specify acceleration of the fluid in the flow component due to body forces such as gravity or centrifugal force.

**Body Force Acceleration (along pipe axis)**

The value entered here will impose an acceleration term on the fluid along the pipe direction, defined by the link (arrow) direction.

**Pressure Loss Coefficients (Bend and Taper Losses for Pipes)**

☉ **Determine Loss Coefficients (Fwd and Rev) from Geometry**

For each pipe in the flow system, calculates the pressure loss coefficient over the length of the pipe automatically based on the geometry (i.e. tapers and bends).

☉ **Zero Pressure Losses from Bends and Tapers**

Neglects all pressure drop effects due to bends and tapers for all pipes in the flow system.





## GEMFlowsplit - General Flowsplit

This template is used to model a general flowsplit that will not be discretized further. It can have any number of inlet or outlet ports.

### Geometry

#### Wall Thickness

Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a '[WallThermalProperty](#)' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the '[WallThermalProperty](#)' reference object.

#### End Type

This attribute specifies the type of ends that will be used when making the flowsplit. The choices and their descriptions include:

- **Closed** indicates that both ends of the flowsplit will be closed.
- **Open** indicates that both ends will be left open and will act as ports.
- **First Cross Section Open** indicates that only the first cross section will be left open to act as a port. The last cross section will be closed.
- **Last Cross Section Open** indicates that only the last cross section will be left open to act as a port. The first cross section will be closed.

### Cross Sections

#### Cross Section Name

Name of the cross section object(s) describing the shape to be used for the flowsplit. The flowsplit allows any of the 4 standard cross section shapes ([circle](#), [ellipse](#), [rectangle](#), [rounded rectangle](#)) as well as the [bi-radial](#) shape. If only 1 cross section is specified, then that cross section will be used at the beginning and end of the flowsplit. If 2 or more are specified, then the first one will be used at the beginning of the flowsplit and the last one will be used at the end. The rest will be placed according to the **Distance to Next** below. The resulting flowsplit will use a smooth transition to change cross sections along the length of the flowsplit.

#### Distance to Next Cross Section

Distance to the next cross section. The cross section specified above in the same column will be extruded for this distance and blended with the next specified cross section. Only a single cross section is necessary, in which case this value will be the total length. When multiple cross sections are specified, this is the length of each section. Also with multiple cross sections, the value of the last distance will be ignored as there is not another cross section to extrude to.

### Location

#### Location X

Specifies the absolute X location of the component's first cross section in







the [Global Coordinate System](#).

<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z</b>	Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.

## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

<b>Surface Area</b>	The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:
---------------------	--





$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

*Area* = **Surface Area**

*D* = **Diameter** (specified above)

*L* = **Length** (specified above)

*D<sub>orifice</sub>* = orifice diameters adjacent to each port

#### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

#### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

#### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

##### Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

#### ☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

#### Additional Geometry



Specify optional geometric characteristics of the flow component.



## Options

### No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

#### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

#### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

#### ☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

#### ☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

#### ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

#### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

#### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and





this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

### Additional Thermal Options

Specify optional thermal characteristics of the flow component.

#### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

#### Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

#### Thermocouple Object

Name of a ['Thermocouple'](#) reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

#### ☉ Heat Transfer Correlation (Colburn)

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

#### ☉ User Defined Heat Transfer Model

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

#### ☉ Heat Transfer Coefficient

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

#### Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off:** No condensation or evaporation is modeled.
- **on\_gas:** Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall:** Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because





the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '**EjectorConn**' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '**SensorConn**' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.





**User Defined Friction  
Model**

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.





## GEMFsplitGeneral - General Flowsplit

This template is used to model a flowsplit created from an imported geometry file. This component will be discretized as a single general flowsplit in the GT-SUITE model. This component can have any number of inlet or outlet ports.

### Boundary Data

<b>Link ID Number</b>	User defined identifier for the mesh flowsplit's ports. Each port number must be a unique positive integer.
<b>Angle wrt X-axis</b>	Angle of the adjacent flow component with respect to the local X-axis. This is the angle that was calculated from the converted original mesh shape. This attribute is read-only.
<b>Override Angle wrt X-axis</b>	Override value for the angle of the adjacent flow component with respect to the local X-axis. This attribute can be used to override the calculated value of this attribute above. A value of "def" specifies that the calculated value will be used for this component during discretization.
<b>Angle wrt Y-axis</b>	Angle of the adjacent flow component with respect to the local Y-axis. This is the angle that was calculated from the converted original mesh shape. This attribute is read-only.
<b>Override Angle wrt Y-axis</b>	Override value for the angle of the adjacent flow component with respect to the local Y-axis. This attribute can be used to override the calculated value of this attribute above. A value of "def" specifies that the calculated value will be used for this component during discretization.
<b>Angle wrt Z-axis</b>	Angle of the adjacent flow component with respect to the local Z-axis. This is the angle that was calculated from the converted original mesh shape. This attribute is read-only.
<b>Override Angle wrt Z-axis</b>	Override value for the angle of the adjacent flow component with respect to the local Z-axis. This attribute can be used to override the calculated value of this attribute above. A value of "def" specifies that the calculated value will be used for this component during discretization.
<b>Characteristic Length</b>	Characteristic flowsplit length facing this port. This is the length that was calculated from the converted original mesh shape. This length is calculated as the perpendicular distance from the associated port to a physical wall or another port. This attribute is read-only.
<b>Override Characteristic Length</b>	Override value for the characteristic flowsplit length facing this port. This attribute can be used to override the calculated value of this attribute above. A value of "def" specifies that the calculated value will be used for this component during discretization.
<b>Actual Port Cross Sectional Area</b>	For reference only, displays the cross-section area at the port (often, this is where the "slice" was made in the solid shape)
<b>Equivalent Diameter (Based on Area)</b>	For reference only, diameter of a circle with area equivalent to the <b>Actual Port Cross Sectional Area</b> .
<b>Expansion Diameter</b>	Expansion diameter of the flowsplit at this port. This is the diameter that





was calculated from the converted original mesh shape. The expansion diameter ( $D_{exp}$ ) is calculated from the characteristic length ( $L_{Char}$ ) of the associated port and the total volume ( $V$ ) assuming a cylindrical shape using the following formula.

$$D_{exp} = \sqrt{\frac{4 \cdot V}{\pi \cdot L_{Char}}}$$

This attribute is read-only.

#### Override Expansion Diameter

Override value for the expansion diameter of the flowsplit at this port. This attribute can be used to override the calculated value of this attribute above. A value of "def" specifies that the calculated value will be used for this component during discretization.

#### Hydraulic Diameter

For reference only, the calculated hydraulic diameter ( $=4 \cdot \text{Area} / \text{Perimeter}$ ) of the port (often, this is where the "slice" was made in the solid shape)

#### User Characteristic Dimension

Only available when the **Flowsplit HTC Calculation Method** in the Thermal folder has been set to "Method #2" using "User\_Dimension". This value will be the Characteristic dimension used in calculations related to the flowsplit's heat transfer coefficient. The value will be either the Equivalent Diameter (Based on Area) or the Hydraulic Diameter, depending on the setting of the option in **File → Options → Conversion → Method for determining User Characteristic Dimension for GEMFSplitGeneral Ports**:

#### Override User Characteristic Dimension

Override value for the **User Characteristic Dimension** of the flowsplit at this port. This attribute can be used to override the calculated value of this attribute above. A value of "def" specifies that the calculated value will be used for this component during discretization.

### Visual

#### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in **File → Options → General**.

#### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black







## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

#### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

#### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value** : 0.0





### ☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

### Additional Geometry Options

Specify optional geometric characteristics of the flow component.

### No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### (☉) Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

### (☉) Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

### (☉) Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part.





## Primitive

The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

## ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

Specify optional thermal characteristics of the flow component.

## Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

## Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

## Thermocouple Object

Name of a 'Thermocouple' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

## ☉ Heat Transfer Correlation (Colburn)

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

## ☉ User Defined Heat Transfer Model

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the 'UserModel' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

## ☉ Heat Transfer Coefficient

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

## Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to 'FluidRefrigerant' Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat





is released entirely to the fluid.

- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

### Flowsplit HTC Calculation Method

#### Method #1 (v7.5 and prior)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, and then they are weighted by the boundary area. This method tends to under predict the heat transfer coefficient in the flowsplit, and is only recommended to reproduce previous version results.



**Method #2  
(Recommended)**

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, which can be based on the expansion diameter or a custom dimension. An enhancement factor based on the flowsplit characteristic length is then added to each port value, and then the resulting coefficients are averaged based on the boundary flow rates. This method produces a more accurate prediction for the heat transfer coefficient, and is recommended especially in applications where the heat transfer in the flowsplit is a significant part of the overall circuit heat transfer rate.

**Characteristic  
Dimension for Port HTC**

This attribute will only be used when Method #2 is selected. The attribute selects which value will be used as the diameter in the heat transfer coefficient calculation. This can either be the Expansion Diameter, which will also be used for the pressure loss calculation, or the User Dimension which can be defined in the Boundary Data for each port. The User Dimensions will only affect the heat transfer coefficient calculation; it will not be used in the pressure loss calculation.

**Pressure Drop****Friction Options**

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

**☉ Friction Multiplier**

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

**☉ No Friction Pressure  
Losses**

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

**User Defined Friction  
Model**

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.



## GEMInertia3D - Mechanical Rigid Body

This template is used to convert a CAD shape/body into a mechanical rigid body which can be exported to GT-ISE as an Inertia3D object. This component automatically calculates the mass and inertia properties of the component. Additional nodes can also be defined on the rigid body which may serve as connection points for other components.

### Material

**Material Object** The name of 'MaterialMechanical' object describing the material properties of the component. This object will be used to calculate the mass and inertia properties. Standard materials are available in the template library, and they can be accessed by right-clicking and selecting "Value Selector".

### Reference Node

☒ **Reference Node Label** A label describing the reference node (port 0). This label will be used in the names of plots and data sets in GT-POST and will also be displayed when linking to the 'Inertia3D'.

**LocalOrigin defining the Reference Node (ign=Global)** Name of the 'LocalOrigin' object describing the reference frame in which the component's initial states are defined. If this attribute is set to "ign", the global (world) coordinate system of the CAD model is used as the reference frame. Note that the coordinate system described in the 'LocalOrigin' object can be imported along with the original CAD model or created in GEM3D.

**Initial X Velocity** Initial X-component of the translational velocity of Reference Node with respect to the CAD Global Origin. ("def" = 0.0)

**Initial Y Velocity** Initial Y-component of the translational velocity of Reference Node with respect to the CAD Global Origin. ("def" = 0.0)

**Initial Z Velocity** Initial Z-component of the translational velocity of Reference Node with respect to the CAD Global Origin. ("def" = 0.0)

**Initial Angular Velocity (X)** Initial X-component of the angular velocity of the body with respect to the CAD Global Origin. ("def" = 0.0)

**Initial Angular Velocity (Y)** Initial Y-component of the angular velocity of the body with respect to the CAD Global Origin. ("def" = 0.0)

**Initial Angular Velocity (Z)** Initial Z-component of the angular velocity of the body with respect to the CAD Global Origin. ("def" = 0.0)

**Contact Geometry Object** Name of the 'ContactGeom3D\*' object describing the surface associated with the rigid body at its Reference Node (port 0) for contact analysis purposes. If this attribute is set to "ign", no surface will be specified.



## Additional Nodes

The Additional Nodes folder specifies the locations of additional nodes present in the rigid body, see Figure 1. These nodes can be used to connect the rigid body to other components or simply to plot the states of the material point associated with the node.

The number of columns that must be defined in this folder = # of unique ports excluding Reference Node (port 0).

### ☒ Node Label

A label describing each of the nodes. This label will be used in the names of plots and data sets in GT-POST and will also be displayed when linking to the 'Inertia3D' component.

### LocalOrigin defining the Additional Node

Name of the '[LocalOrigin](#)' object describing the local reference frame for the additional node. Note that the coordinate system described in the '[LocalOrigin](#)' object can be imported along with the original CAD model or created in GEM3D.

### Export Nodal Position with respect to

Two options are available for exporting the nodal position information of the local origin selected above:

- **Reference Node** When this option is selected, the coordinates of the additional Node are defined with respect to the local reference node as defined in the Reference Node tab.
- **Global Origin** When this option is selected, the coordinates of the additional Node are defined with respect to the CAD global (world) coordinate system.

### Contact Geometry Object

Name of the '[ContactGeom3D\\*](#)' object describing the surface associated with the rigid body at its additional node Node # for contact analysis purposes. If this attribute is set to "ign", no surface will be specified.

## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black



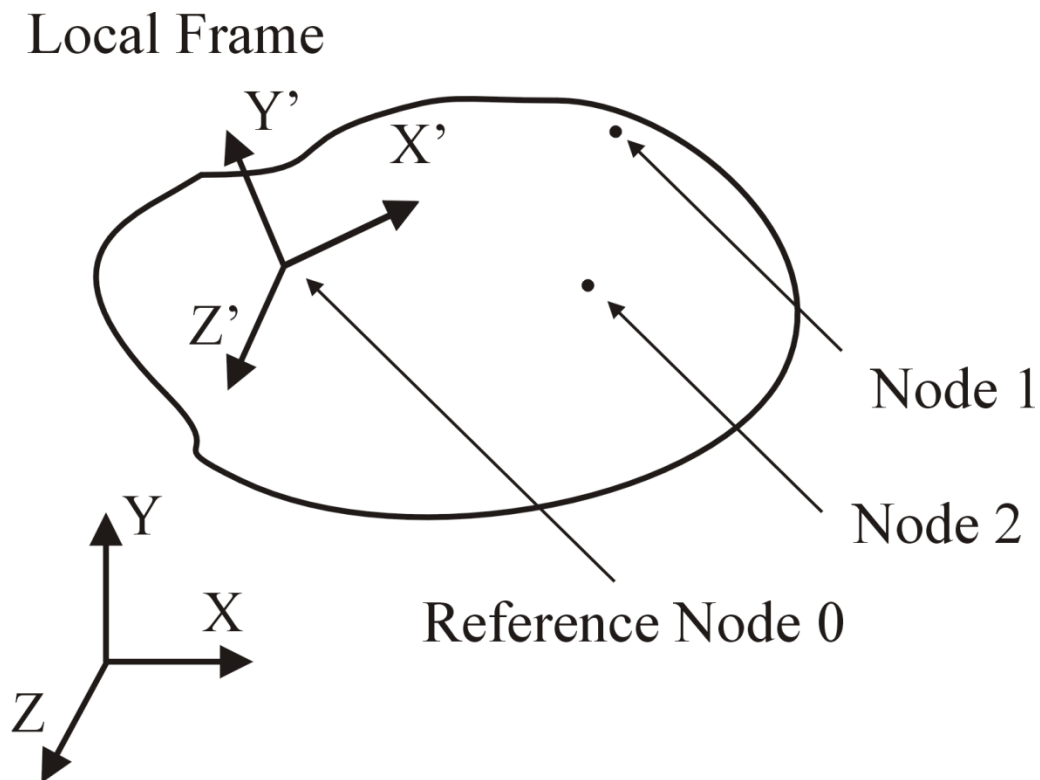


Figure 1. Rigid body configuration: Definition of its local frame and additional nodes.







## GEMMechSolid3D – 3D Finite Element Solid Body

This template is used to represent a mechanical body using a 3D finite element component meshed from a 3D solid geometry. GEM3D includes a built-in meshing tool which can generate this component when accessed via the standard “Convert Shape to Component” function. Alternatively, this component is also created when converting an external imported 3D mesh ([GEMFEVolumeMesh](#)) to a GEM FE component.

### Main

#### FE Mesh Reference Object

The name of a [FEMesh3D](#) reference object which contains all of the node and element data for the mesh. This object is generated automatically by GEM3D during the meshing operation or the conversion of an external imported 3D mesh. Note that some of the data in this ref. object is completed by GEM3D upon export of the GT model file based on the presence of children FE ports.

#### Material Properties Object (for all FE Elements)

The name of a [MaterialMechanical](#) reference object which contains the material properties of the FE mesh. These properties will be assigned to all elements in the mesh.

Note: After exporting the model, it is possible to manually edit the resulting objects to define multiple materials if using the ‘[Solid3D](#)’ template.

### Visual

#### Transparency Percent

Indicates the transparency level used when drawing the component. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

#### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black





## **GEMMeshShape - General Mesh Shape**

This template is used to represent a shape based on surface geometry using individual triangles. Mesh shapes may be imported from STL or IGES files. This component cannot and will not be discretized into an exported model file (.gtm). To make sure this component is represented when discretized, it must first be converted into a GEM3D component using the [Convert Shape to Component](#) operation. The conversion operation can be undone using the [De-Convert Mesh](#) operation as long as the "Allow Mesh Deconversion" option is turned on (checked) in [File>Options>Conversion - Contains the conversion preferences](#).

---

### **Visual**

---

#### **Transparency Percent**

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

#### **Display Color**

Indicates the color used when drawing the component. The color choices include:

- **Red**
- **Blue**
- **Dark Blue**
- **Green**
- **Copper**
- **Gold**
- **Grey**
- **Black**





## GEMMeshShell - Mesh Shell

This template is used to model a general volume constructed of small triangular surfaces formed by an [external geometry file](#) that will be discretized into many smaller flowsplits.

### Orientation

*The attributes on this tab specify the location and orientation of the [local coordinate system \(LCS\)](#) that will be used as the reference during feature creation (baffles, etc.) and discretization. When it is necessary to change the location and orientation of the component's LCS, it is recommended to use the apply button in the component often to see a visual update. When doing this the [Component's Axis](#) should be shown, which can be done from the right-click menu for the component.*

<b>X Location of LCS</b>	Specifies the absolute X location of the component's LCS in the <a href="#">Global Coordinate System</a> .
<b>Y Location of LCS</b>	Specifies the absolute Y location of the component's LCS in the <a href="#">Global Coordinate System</a> .
<b>Z Location of LCS</b>	Specifies the absolute Z location of the component's LCS in the <a href="#">Global Coordinate System</a> .
<b>X Direction of LCS</b>	Specifies the X component of the vector describing the direction that the component's major axis will be pointed. Only a unit vector is needed to describe the direction, so the <b>X Direction of LCS</b> , <b>Y Direction of LCS</b> , and <b>Z Direction of LCS</b> attributes may be replaced with the equivalent unit vector.
<b>Y Direction of LCS</b>	Specifies the Y component of the vector describing the direction that the component's major axis will be pointed. Only a unit vector is needed to describe the direction, so the <b>X Direction of LCS</b> , <b>Y Direction of LCS</b> , and <b>Z Direction of LCS</b> attributes may be replaced with the equivalent unit vector.
<b>Z Direction of LCS</b>	Specifies the Z component of the vector describing the direction that the component's major axis will be pointed. Only a unit vector is needed to describe the direction, so the <b>X Direction of LCS</b> , <b>Y Direction of LCS</b> , and <b>Z Direction of LCS</b> attributes may be replaced with the equivalent unit vector.
<b>Reference Direction Angle</b>	Specifies the rotational orientation of the LCS. The angle specified here is used to rotate the orientation of the LCS from the default calculated values given by the following 3 reference direction attributes. A good practice is to show the component's axis in the graphical window (right-click on component→Component's Axis) and modify this angle until the desired orientation is reached (use the Apply button in the component to visually see the changes).
<b>Reference Direction X</b>	Specifies the X component of the vector describing the reference direction. The reference direction gives the direction of the X axis of the LCS. It is highly recommended to use a value of "def", letting the application determine this value. When using "def", it must be used for





all reference direction attributes (X, Y, and Z) together (or none).

*Only a unit vector is needed to describe the reference direction, so the **Reference Direction X**, **Reference Direction Y**, and **Reference Direction Z** attributes may be replaced with the equivalent unit vector.*

#### Reference Direction Y

Specifies the Y component of the vector describing the reference direction. The reference direction gives the direction of the X axis of the LCS. It is highly recommended to use a value of "def", letting the application determine this value. When using "def", it must be used for all reference direction attributes (X, Y, and Z) together (or none).

*Only a unit vector is needed to describe the reference direction, so the **Reference Direction X**, **Reference Direction Y**, and **Reference Direction Z** attributes may be replaced with the equivalent unit vector.*

#### Reference Direction Z

Specifies the Z component of the vector describing the reference direction. The reference direction gives the direction of the X axis of the LCS. It is highly recommended to use a value of "def", letting the application determine this value. When using "def", it must be used for all reference direction attributes (X, Y, and Z) together (or none).

*Only a unit vector is needed to describe the reference direction, so the **Reference Direction X**, **Reference Direction Y**, and **Reference Direction Z** attributes may be replaced with the equivalent unit vector.*

## Discretization

#### Discretization Scheme

Specifies which discretization values to use for this shell. The choices include:

- **Use Global Values** indicates the discretization values for this shell will be taken from the global discretization values specified in the [Export gtm](#) command.
- **Use Values Below** indicates the discretization values will be taken from the attributes below for this shell. If selected, then values specified below will override the global values specified in the [Export gtm](#) command. A value of "def" for any of the attributes below will use the global value.

#### Shell Discretization Length Override Along Local X

Specifies the override target discretization length in the local X direction (X direction of the shell's [local coordinate system](#)) for this mesh shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

#### Shell Discretization Length Override Along Local Y

Specifies the override target discretization length in the local Y direction (Y direction of the shell's [local coordinate system](#)) for this mesh shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.





### Shell Discretization Length Override Along Local Z

Specifies the override target discretization length in the local Z direction (Z direction of the shell's [local coordinate system](#)) for this mesh shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

### Acceptance Ratio Override

Specifies the override flowsplit acceptance ratio for this mesh shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Vertices

*The mesh shell forms the shape of the component using the geometry of each triangle (similar to the STL file format). The attributes in this folder specify information related to the vertices points of each triangle (3 per triangle). The data cannot be changed manually. It is only stored as a convenience to copy the points if necessary.*

### IDV

Specifies the vertices ID number (assigned randomly).

### X

Specifies the absolute X location of the vertices point in the [Global Coordinate System](#).

### Y

Specifies the absolute Y location of the vertices point in the [Global Coordinate System](#).

### Z

Specifies the absolute Z location of the vertices point in the [Global Coordinate System](#).





## Triangles

The mesh shell forms the shape of the component using the geometry of each triangle (similar to the STL file format). The attributes in this folder specify information related to the triangles. The data cannot be changed manually. It is only stored as a convenience to copy the points if necessary.

<b>IDT</b>	Specifies the triangle ID number (assigned randomly).
<b>V1</b>	Specifies the vertex point ID number (IDV in previous folder) of the first vertex point of the triangle.
<b>V2</b>	Specifies the vertex point ID number (IDV in previous folder) of the second vertex point of the triangle.
<b>V3</b>	Specifies the vertex point ID number (IDV in previous folder) of the third vertex point of the triangle.
<b>Port Info</b>	Specifies a value if the triangle borders on an open port of the mesh shell. Triangles with the same value mean they are a part of the same port. Specifies 0 if it borders other triangles.

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

#### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### © Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).





## ◎ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

## ◎ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

### Additional Geometry Options

Specify optional geometric characteristics of the flow component.

## No. of Identical Flowsplits

Number of identical, parallel flowsplits represented by this individual flowsplit (usually 1.0, except for modeling many small flowsplits inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

## ◎ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall





## ☉ Calculated Wall Temperature

temperature distribution, where X is normalized length with range 0 to 1.

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

## ☉ Wall Layer Properties Object

Name of the '[WallThermalProperty](#)' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

## ☉ Wall External Boundary Conditions Object

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

## ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

## ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

## ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

Specify optional thermal characteristics of the flow component.

## Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

## Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

## Thermocouple Object

Name of a '[Thermocouple](#)' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.







## User Defined Heat Transfer Model

The name of the **'UserModel'** object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

## Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to **'FluidRefrigerant'** Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an **'EjectorConn'** object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity **'SensorConn'** output signal. If a study is particularly sensitive





to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

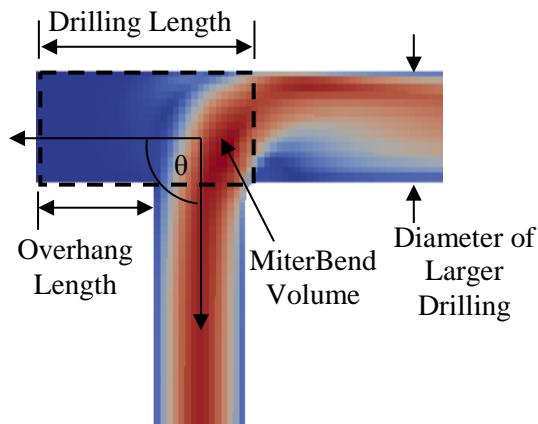
#### User Defined Friction Model

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.



GEMMiterBend – Miter Bend

This template is used to model a miter bend and represents the volume in a sharp change in flow direction at the intersection of two flow passages, similar to the configuration below. The volume of the intersection is accounted for, along with the pressure loss associated with the sudden change in flow direction based on the miter angle. Friction is also included. Any overhang volume due to over-drilling will also be accounted for.



Visual

Transparency Percent	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
Display Color	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>

Main

Basic Geometry and Initial Conditions	Specify required geometric input as well as the initial state of the fluid in the flow component.
Diameter of Larger Drilling	The diameter of the larger of the two intersecting drillings. See the schematic above for a visual representation.
Drilling Length (includes	The length of the drilling which usually can be calculated as the diameter



overhang)	of the shorter drilling plus the length of the overhang. See the schematic above for a visual representation.
Angle of Miter Bend	The angle of the miter bend, as represented by $\theta$ in the schematic above. A value of zero means there is no change in direction.
Initial State Name	Name of the 'FluidInitialState' reference object describing the initial conditions inside the miter bend.
<b>Surface Finish</b>	Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.
☉ Smooth	Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).
☉ Roughness from Material	This attribute gives choices of materials that may be used to specify the surface roughness.  <u>Material Name : Default Roughness Value (mm)</u> <ul style="list-style-type: none"> <li>• drawn_metal : 0.002</li> <li>• steel : 0.046</li> <li>• cast_iron : 0.26</li> <li>• light_rust_steel : 0.25</li> <li>• heavy_rust_steel : 1.0</li> <li>• smooth_plastic : 0.0025</li> <li>• smooth_rubber : 0.025</li> <li>• smooth_galvanized : 0.025</li> <li>• normal_galvanized : 0.15</li> <li>• wrought_iron : 0.046</li> <li>• asphalted_cast : 0.12</li> <li>• extruded_aluminum : 0.003</li> <li>• user_value: 0.0</li> </ul>
☉ Sand Roughness	Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.  A numeric value or a parameter may be entered for this attribute.

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to



☉ Imposed Wall Temperature

ambient.

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses). This option should be used when this Pipe object will be an intake or exhaust port for a cylinder.

☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

Additional Thermal Options

Specify optional thermal characteristics of the flow component.

Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of



Run Setup for information about the global "Heat Transfer Enhancement Factor")

## Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

## ☉ Heat Transfer Correlation (Colburn)

Calculates the heat transfer coefficient using the Colburn analogy. For more information please see Colburn analogy in Flow Theory Manual.

## ☉ User Defined Heat Transfer Model

The name of the 'UserModel' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

## ☉ Heat Transfer Coefficient

Imposes a flow heat transfer coefficient between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** and **Global Heat Transfer Multiplier** will be ignored.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

## ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

## ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

## Loss Coefficient Multiplier

A multiplier to the internally calculated loss coefficient vs. Reynolds number and bend angle. Note this multiplier is applied separately from the friction multiplier, thus giving the user control over both the friction component and geometrical component of loss via separate multipliers.





## GEMMultiplePipe - Bundle of Multiple Small Pipes

This template is used to model a component that has a bundle of smaller pipes. This component can be used to model coolers and other similar parts that have many small flow paths. Multiple pipe components will be drawn with an X in the center of the cross section to differentiate it from other pipes in the graphical window.

### Geometry

<b>Number of Pipes</b>	Total number of pipes in the bundle.
<b>Wall Thickness</b>	Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a <a href="#">'WallThermalProperty'</a> reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the <a href="#">'WallThermalProperty'</a> reference object.

### Cross Sections

<b>Cross Section Name</b>	Name of the cross section object(s) describing the shape to be used for a <u>single</u> pipe. The total size of the bundle will be determined from this cross section and knowledge of the <b>Number of Pipes</b> specified above. Any of the 4 standard cross section shapes (circle, ellipse, rectangle, rounded rectangle) are allowed. If only 1 cross section is specified, then that cross section will be used at the beginning and end of the pipe. If 2 or more are specified, then the first one will be used at the beginning of the pipe and the last one will be used at the end. The resulting pipe will use a smooth transition to change cross sections along the length of the pipe.
<b>Distance to Next Cross Section</b>	Distance to the next cross section. The cross section specified above in the same column will be extruded for this distance and blended with the next specified cross section. Only a single cross section is necessary, in which case this value will be the pipe length. When multiple cross sections are specified, this is the length of each section. Also with multiple cross sections, the value of the last distance will be ignored as there is not another cross section to extrude to.

### Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the





extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

#### Direction Y

Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

#### Direction Z

Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

### Visual

#### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

#### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

### Main

#### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Discretization Length

Discretization length to be used for pipes during the discretization process. This length does not need to be an even fraction of the entire pipe length; the code will adjust to divide the pipe appropriately. A value of "def" will use the discretization length found in the global discretization window. See [export gtm](#) for additional information.

#### Initial State Name

Name of the '[FluidInitialState](#)' reference object describing the initial conditions inside the pipe.

#### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.







☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

**Additional Geometry Options**

Specify optional geometric characteristics of the flow component.

**Pipe Elevation Change**

This attribute defines the elevation change of the pipe from the inlet to the outlet. A positive value means port 2 is at a higher elevation than port 1, and vice versa. If "def" is entered, then the elevation change of the pipe will be automatically calculated from the options selected in the **Gravitational Vector** field of [File>Options>Discretization - Contains the discretization preferences](#).

This attribute cannot be used in parallel with the **Body Force Acceleration** attribute under the Options folder. The acceleration due to gravity is assumed to be  $9.80665 \text{ m/s}^2$ . When 'XYTable' reference object is used, X is defined as pipe length (which is normalized) and Y is the elevation change.

**Thermal**

**Wall Temperature**



Specify the thermal characteristics of the flow component. In many



## Method

### ☉ Imposed Wall Temperature

cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

#### (☉) Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

#### (☉) Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

#### (☉) Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses). This option should be used when this Pipe object will be an intake or exhaust port for a cylinder.

### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

### Heat Transfer Multiplier



Specify optional thermal characteristics of the flow component.

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-



	smooth option is selected for <b>Surface Finish</b> , additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")
<b>Heat Input Rate</b>	The rate of heat input to the fluid or the name of a dependency reference object.
<b>Thermocouple Object</b>	Name of a ' <a href="#">Thermocouple</a> ' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe* or FlowSplit* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.
<b>☉ Heat Transfer Correlation (Colburn)</b>	Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.
<b>☉ User Defined Heat Transfer Model</b>	Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to defined the name of the ' <a href="#">UserModel</a> ' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the <b>Heat Transfer Multiplier</b> value.
<b>☉ Heat Transfer Coefficient</b>	Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.
<b>Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)</b>	<p>Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to '<a href="#">FluidRefrigerant</a>' Circuits, which do not require any settings to enable boiling/condensation.</p> <ul style="list-style-type: none"> <li>• <b>off</b>: No condensation or evaporation is modeled.</li> <li>• <b>on_gas</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.</li> <li>• <b>on_wall</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.</li> </ul> <p>Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as <b>off</b> and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also</p>





associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

#### ☉ Friction Multiplier

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the '[UserModel](#)' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

### Acceleration Options

Specify acceleration of the fluid in the flow component due to body forces such as gravity or centrifugal force.





**Body Force Acceleration  
(along pipe axis)**

The value entered here will impose an acceleration term on the fluid along the pipe direction, which is defined by the link (arrow) direction.

**Pressure Loss  
Coefficients (Bend and  
Taper Losses)**

Specify the influences of the bends and taper losses on the Pipe pressure drop. Please note that choosing any option other than **Determine Loss Coefficients (Fwd and Rev) from Geometry** will ignore the influence of user-specified bends and tapers in pressure drop calculations.

**☉ Determine Loss  
Coefficients (Fwd and  
Rev) from Geometry**

Calculates the pressure loss coefficient over the length of the pipe automatically based on the geometry (i.e. tapers and bends).

**(☉) Zero Pressure  
Losses from Bends and  
Tapers**

Neglects all pressure drop effects due to bends and tapers. This is equivalent to specifying **Forward Loss Coefficient** and **Backward Loss Coefficient** as zero. Note that friction losses are computed separately.

**(☉) Forward Pressure  
Loss Coefficient**

Pressure loss coefficient for flow from port 1 to port 2. This attribute is used to account for pressure losses due to geometry, such as bends and non-circular cross-sections. Enter "def" to have the pressure loss automatically calculated by the code using the cross-sectional shape and bend information that is entered. (0 indicates no additional pressure loss.)

**(☉) Reverse Pressure  
Loss Coefficient**

Pressure loss coefficient in the opposite direction from port 2 to port 1 (see **Forward Pressure Loss Coefficient** above).





## GEMOrifice - Orifice on the Surface of a Shell

This template is used to add an orifice to the surface of a shell. The orifice must be circular and located on the surface of the shell. The orifice will be created by extruding a cylinder through the shell and removing the intersected portion. The cylinder will have a diameter equal to the diameter specified below and be extruded from the specified location in the specified direction.

### Geometry

<b>Location X (Local)</b>	Specifies the X location on the shell of the center of the orifice. This location is relative to the shell's <a href="#">local components axis</a> .
<b>Location Y (Local)</b>	Specifies the Y location on the shell of the center of the orifice. This location is relative to the shell's <a href="#">local components axis</a> .
<b>Location Z (Local)</b>	Specifies the Z location on the shell of the center of the orifice. This location is relative to the shell's <a href="#">local components axis</a> .
<b>Direction X (Local)</b>	Specifies the X component of the vector describing the direction that the orifice will be created. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y (Local)</b>	Specifies the Y component of the vector describing the direction that the orifice will be created. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z (Local)</b>	Specifies the Z component of the vector describing the direction that the orifice will be created. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Diameter</b>	Orifice diameter.

### Main

<b>Forward Discharge Coefficient</b>	Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.
<b>Reverse Discharge Coefficient</b>	Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Options

<b>Forward End Correction (Length/Diameter)</b>	End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change.
---	--





## Reverse End Correction (Length/Diameter)

This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

## Heat Conduction "Flange"

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

## Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one '[OrificeConn](#)' connection in a strong of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def" The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery



This attribute controls the method of predicting the pressure loss across





## Choice

the connection :

- **PressureRecovery**

This option enables the pressure recovery in Orifice Connections as described in the Flow Manual, Chapter 2.2.8. If this option is used and discharge coefficients are specified, the resulting pressure drop may not be equal to Bernoulli's Law due to the pressure recovery calculated. **This is the default value which has been used by GT-Power in all previous versions.**

- **NoRecovery**

This option disables the pressure recovery in the Orifice Connection. With this choice, the pressure drop across the Orifice Connection is calculated according to Bernoulli's Law.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def".

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is not allowed.







## GEMOrificeBaffle - Orifice through a Baffle

This template is used to model an orifice through a baffle. This can be done using the template attributes at the top of the window and the 2D graphical interface at the bottom of the window. This editor is used to choose and position a cross section on the baffle to be used as an orifice. The graphical window will automatically contain the shape of the baffle. This allows the perforated cross section to be placed anywhere on the baffle by placing it in the desired location in the graphical window. An existing or new cross section can be added to the graphical window to use it as the orifice shape. This section can then be placed interactively in the graphical window to position the orifice or the template location attributes may be used. The interface in the graphical window is essentially the same as the [Cross Section Editor](#), however, with some restricted capabilities.

### Geometry

<b>Cross Section</b>	Name of the cross section object describing the shape to be used for the orifice. Any of the 4 standard cross section shapes ( <a href="#">circle</a> , <a href="#">ellipse</a> , <a href="#">rectangle</a> , <a href="#">rounded rectangle</a> ) as well as a <a href="#">custom shape</a> may be used. Alternatively the <b>Add Cross Section</b> option from the graphical window interface below the template window may be used (see commands after this table).
<b>Location X</b>	Specifies the X location on the baffle of the center of the orifice. This location is relative to the <a href="#">local components axis</a> .
<b>Location Y</b>	Specifies the Y location on the baffle of the center of the orifice. This location is relative to the <a href="#">local components axis</a> .

### Main

<b>Forward Discharge Coefficient</b>	Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.
<b>Reverse Discharge Coefficient</b>	Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Options

<b>Forward End Correction (Length/Diameter)</b>	End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very
---	--





### **Reverse End Correction (Length/Diameter)**

strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### **Heat Conduction "Flange"**

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

### **Initial Mass Flow Rate**

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one '[OrificeConn](#)' connection in a strong of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def" The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

### **Pressure Recovery Choice**

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery**

This option enables the pressure recovery in Orifice Connections as described in the Flow Manual, Chapter 2.2.8. If this option is used and discharge coefficients are specified, the resulting pressure drop





may not be equal to Bernoulli's Law due to the pressure recovery calculated. **This is the default value which has been used by GT-Power in all previous versions.**

- **NoRecovery**

This option disables the pressure recovery in the Orifice Connection. With this choice, the pressure drop across the Orifice Connection is calculated according to Bernoulli's Law.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def".

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is not allowed.

The commands used in the graphical window to place the orifice cross sections are available on the toolbar above the graphical window. They are described below.



**Undo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row.



**Redo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row.



**Toggle Snap to Grid:** Toggles the snap to grid function. With this enabled, all locations will snap to the nearest grid point.



**Toggle Grid:** Toggles the display of the grid.



**Zoom In:** Zooms in on the cross section. [Keyboard hotkey = NumPad +]



**Zoom Out:** Zooms out from the cross section. [Keyboard hotkey = NumPad -]



**Zoom 1:1:** Restores the zoom to the default level (1:1).



**Fit to Screen:** Centers the cross section and zooms to the appropriate level to fit the entire cross section to the editor window.



**Center Canvas:** Centers the canvas on the origin without changing the zoom level.





**Unit:** Drop down menu specifying the units for the cross sections. Choices include m, cm, mm, in, ft, micron, km, mi, and dm.



**Rotate:** Rotates the cross section used as the orifice section in the plane of the parent baffle.



**Add Cross Section:** Adds an existing cross section or allows the creation of a new cross section to be used as the perforate section. This will open the cross section manager window that will allow selection of any existing cross sections or the creation of new ones.



**Delete Cross Section:** Deletes the current cross section from the graphical window only. This means that the perforate section is deleted, not that the actual cross section is deleted. The actual cross section will still be available in the model.



**Configure Ruler:** Allows the ruler spacing to be manually specified.





## GEMOrificePipe - Internal Orifice in a Straight Pipe

This template is used to add an internal orifice to a round pipe or straight pipe.

### Geometry

<b>Orifice Diameter</b>	Orifice diameter.
<b>Hole Thickness</b>	Thickness of the orifice plate.
<b>Location Reference</b>	<p>Specifies the reference location of the component in which to position the internal orifice. Choices include:</p> <ul style="list-style-type: none"><li>• <b>From start</b> indicates that the <b>Distance from Reference</b> value below specifies the position of the internal orifice from the start of the pipe (first cross section).</li><li>• <b>From end</b> indicates that the <b>Distance from Reference</b> value below specifies the position of the internal orifice from the end of the pipe (last cross section).</li></ul>
<b>Distance from Reference</b>	Specifies the distance from the reference along the pipe where the internal orifice will be placed.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	<p>Indicates the color used when drawing the part. The color choices include:</p> <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>

### Main

<b>Forward Discharge Coefficient</b>	Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.
<b>Reverse Discharge</b>	Discharge coefficient in the reverse direction of the linking arrows or the





## Coefficient

name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

## Options

### Forward End Correction (Length/Diameter)

End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Reverse End Correction (Length/Diameter)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Heat Conduction "Flange"

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

### Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in





one 'OrificeConn' connection in a strong of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery**

This option enables the pressure recovery in Orifice Connections as described in the Flow Manual, Chapter 2.2.8. If this option is used and discharge coefficients are specified, the resulting pressure drop may not be equal to Bernoulli's Law due to the pressure recovery calculated. **This is the default value which has been used by GT-Power in all previous versions.**

- **NoRecovery**

This option disables the pressure recovery in the Orifice Connection. With this choice, the pressure drop across the Orifice Connection is calculated according to Bernoulli's Law.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def".

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is not allowed.





## GEMOrificePipeBend - Internal Orifice in a Bent Pipe

This template is used to add an internal orifice to a bent pipe.

### Geometry

<b>Orifice Diameter</b>	Orifice diameter.
<b>Hole Thickness</b>	Thickness of the orifice plate.
<b>Location Reference</b>	<p>Specifies the reference location of the component in which to position the internal orifice. Choices include:</p> <ul style="list-style-type: none"><li>• <b>From start</b> indicates that the <b>Distance from Reference</b> value below specifies the position of the internal orifice from the start of the pipe (first cross section).</li><li>• <b>From end</b> indicates that the <b>Distance from Reference</b> value below specifies the position of the internal orifice from the end of the pipe (last cross section).</li></ul>
<b>Distance from Reference</b>	Specifies the distance from the reference along the pipe where the internal orifice will be placed.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	<p>Indicates the color used when drawing the part. The color choices include:</p> <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>

### Main

<b>Forward Discharge Coefficient</b>	Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.
<b>Reverse Discharge</b>	Discharge coefficient in the reverse direction of the linking arrows or the







**Coefficient**

name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

**Options**

**Forward End Correction  
(Length/Diameter)**

End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

**Reverse End Correction  
(Length/Diameter)**

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

**Heat Conduction  
"Flange"**

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

**Initial Mass Flow Rate**

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in





one 'OrificeConn' connection in a strong of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery**

This option enables the pressure recovery in Orifice Connections as described in the Flow Manual, Chapter 2.2.8. If this option is used and discharge coefficients are specified, the resulting pressure drop may not be equal to Bernoulli's Law due to the pressure recovery calculated. **This is the default value which has been used by GT-Power in all previous versions.**

- **NoRecovery**

This option disables the pressure recovery in the Orifice Connection. With this choice, the pressure drop across the Orifice Connection is calculated according to Bernoulli's Law.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def".

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is not allowed.





## GEMOverPipe - Overlapping Pipes

This template is used to model an overlapping pipe (a pipe that is inside of another pipe). The overlapping pipe is represented by two pipes. Pipe 1 is the primary component and will be the anchor for the location and direction of the overlapping pipe. Pipe 2 will be the secondary component whose position is determined based on the overlapping distance relative to pipe 1. Either pipe 1 or pipe 2 may be the overlapping pipe (have the larger diameter), but the walls cannot intersect.

### Geometry

<b>Pipe 1 Cross Section Name</b>	Name of the cross section object describing the shape to be used for pipe 1. Any of the 4 standard cross section shapes ( <a href="#">circle</a> , <a href="#">ellipse</a> , <a href="#">rectangle</a> , or <a href="#">rounded rectangle</a> ) are allowed.
<b>Pipe 1 Length</b>	Length of pipe 1.
<b>Pipe 1 Wall Thickness</b>	Wall thickness of pipe 1. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a ' <a href="#">WallThermalProperty</a> ' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the ' <a href="#">WallThermalProperty</a> ' reference object.
<b>Pipe 2 Cross Section Name</b>	Name of the cross section object describing the shape to be used for pipe 2. Any of the 4 standard cross section shapes ( <a href="#">circle</a> , <a href="#">ellipse</a> , <a href="#">rectangle</a> , or <a href="#">rounded rectangle</a> ) are allowed.
<b>Pipe 2 Length</b>	Length of pipe 2.
<b>Pipe 2 Wall Thickness</b>	Wall thickness of pipe 2. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a ' <a href="#">WallThermalProperty</a> ' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the ' <a href="#">WallThermalProperty</a> ' reference object.
<b>Length of Overlapping Section</b>	Distance that pipe 1 and pipe 2 overlap.

### Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section (cross section of pipe 1) in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section (cross section of pipe 1) in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section (cross section of pipe 1) in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the





component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

## Direction Z

Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main\_Pipe1 and Main\_Pipe2

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

### Discretization Length

Discretization length to be used for pipes during the discretization process. This length does not need to be an even fraction of the entire pipe length; the code will adjust to divide the pipe appropriately. A value of "def" will use the discretization length found in the global discretization window. See [export gtm](#) for additional information.

### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

### © Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).





### ◎ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

### ◎ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

### Additional Geometry Options

Specify optional geometric characteristics of the flow component.

### Pipe Elevation Change

This attribute defines the elevation change of the pipe from the inlet to the outlet. A positive value means port 2 is at a higher elevation than port 1, and vice versa. If "def" is entered, then the elevation change of the pipe will be automatically calculated from the options selected in the **Gravitational Vector** field of [File>Options>Discretization - Contains the discretization preferences](#).

This attribute cannot be used in parallel with the **Body Force Acceleration** attribute under the Options folder. The acceleration due to gravity is assumed to be  $9.80665 \text{ m/s}^2$ . When 'XYTable' reference object is used, X is defined as pipe length (which is normalized) and Y is the elevation change.

### No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal\_Pipe1 and Thermal\_Pipe2

### Wall Temperature



Specify the thermal characteristics of the flow component. In many



## Method

### ☉ Imposed Wall Temperature

cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

#### (☉) Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

#### (☉) Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

#### (☉) Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

#### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses). This option should be used when this Pipe object will be an intake or exhaust port for a cylinder. This option should be used when this Pipe object will be an intake or exhaust port for a cylinder.

### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

### Heat Transfer Multiplier



Heat transfer coefficient multiplier. The calculated heat transfer rate



## Heat Input Rate

between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

The rate of heat input to the fluid or the name of a dependency reference object.

## Thermocouple Object

Name of a **'Thermocouple'** reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

## User Defined Heat Transfer Model

The name of the **'UserModel'** object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

## Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to **'FluidRefrigerant'** Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor







and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an 'EjectorConn' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity 'SensorConn' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop\_Pipe1 and Pressure Drop\_Pipe2

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

### Acceleration Options

Specify acceleration of the fluid in the flow component due to body forces such as gravity or centrifugal force.

#### Body Force Acceleration (along pipe axis)

The value entered here will impose an acceleration term on the fluid along the pipe direction, which is defined by the link (arrow) direction.

### Pressure Loss Coefficients (Bend and Taper Losses)

Specify the influences of the bends and taper losses on the Pipe pressure drop. Please note that choosing any option other than **Determine Loss Coefficients (Fwd and Rev) from Geometry** will ignore the influence of user-specified bends and tapers in pressure drop calculations.







**☉ Determine Loss Coefficients (Fwd and Rev) from Geometry**

Calculates the pressure loss coefficient over the length of the pipe automatically based on the geometry (i.e. tapers and bends).

**☉ Zero Pressure Losses from Bends and Tapers**

Neglects all pressure drop effects due to bends and tapers. This is equivalent to specifying **Forward Loss Coefficient** and **Backward Loss Coefficient** as zero. Note that friction losses are computed separately.

**☉ Forward Pressure Loss Coefficient**

Pressure loss coefficient for flow from port 1 to port 2. This attribute is used to account for pressure losses due to geometry, such as bends and non-circular cross-sections. Enter "def" to have the pressure loss automatically calculated by the code using the cross-sectional shape and bend information that is entered. (0 indicates no additional pressure loss.)

**(☉) Reverse Pressure Loss Coefficient**

Pressure loss coefficient in the opposite direction from port 2 to port 1 (see **Forward Pressure Loss Coefficient** above).





## GEMParticulateFilter - Wall-Flow Particulate Filter Model

This template is used to model a wall-flow particulate filter for diesel particulate filter (DPF) and gasoline particulate filter (GPF) applications. Please refer to the Aftertreatment Manual for more information about DPF and GPF modeling. Also see the Help section for the 'ParticulateFilter' template for more information. Filter components will be drawn with a diamond in the center of the cross section to differentiate it from other pipes in the graphical window.

### Geometry

<b>Frontal Diameter</b>	The diameter of the substrate. This attribute determines total frontal area of the filter.
<b>Channel Length</b>	Filter channel (cell) length.
<b>Shell Wall Thickness for Visual Effects</b>	Wall thickness of the filter shell for visual effects only. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in the Thermal folder. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the Thermal folder.

### Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z</b>	Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default</b>
-----------------------------	--





**Transparency Percentage** option in File → Options → General.

## Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main

☉ Cell Density	Number of cells (channels) per frontal cross-sectional area.
☉ Inlet Cell Density (inlet cells only)	The number of <u>inlet</u> channels per frontal cross-sectional area. <b>Cell Density</b> divided by 2.
☉ Number of Inlet Channels	The number of <u>inlet</u> channels.
Number of Axial Discretization Intervals	Number of axial discretization intervals, or sub-volumes, along the channel length. Analogous to the discretization length used in other flow components in GT-SUITE, except here the number of sub-volumes is directly specified rather than being calculated. Enter 1 for 0D (lumped model), and a number greater than 1 for an axially discretized 1D model. Recommended value for soot regeneration is the "def" value of 30. For a full-size filter with storage catalyst a value of 50 to 100 is needed. Recommended minimum value for a 1D axially discretized model is 10.
Initial State Object	Name of the <a href="#">FluidInitialState</a> reference object describing the initial conditions of the gas inside the filter.
☉ Substrate Wall Thickness	Substrate wall thickness of the filter, total distance in the radial direction from inlet channel to outlet channel. Refer to figures below.
☉ Inlet Channel Width	The width of a single inlet channel (cell). Refer to figures below.
(☉) Outlet Channel Width	The width of a single outlet channel (cell). Refer to figures below. This value is used to model asymmetric channel width, and must be less than or equal to the <b>Inlet Channel Width</b> . If this attribute is set to "def", the code will set the <b>Outlet Channel Width</b> equal to the <b>Inlet Channel Width</b> .

## Pressure Drop

Clean Filter Wall Permeability	Permeability of a clean filter wall required for pressure drop calculation. This is the primary attribute for calibrating clean filter pressure drop. Values range from 1E-12 to 1E-6 mm <sup>2</sup> , with typical values on the order
--------------------------------	--





of  $1\text{E-}7 \text{ mm}^2$ . If the value is unknown then as a first guess "def" may be entered to have the value calculated from the formula below (Clean Filter Permeability RLT under Initial). Once the calculated value is known, if it gives correct pressure drop then it may be used as is, otherwise it may be used as the middle of the range for calibration.

$$k_{clean} = \frac{\varepsilon^3 d^2}{180 \cdot (1 - \varepsilon)^2}$$

Where  $k_{clean}$  is the **Clean Filter Wall Permeability**,  $\varepsilon$  is the **Filter Porosity**, and  $d$  is the **Pore Diameter**. This formula comes from Porous Media, Dullien, 1979.

**Pore Diameter**

Average diameter of filter substrate pores. Typical values may range from 10-40  $\mu\text{m}$ . "def" = 15  $\mu\text{m}$

**Filter Porosity**

Porosity of filter substrate. This attribute is very important in evaluation of collection efficiency and pressure drop. Typical values range from 0.48 - 0.50, although a value as high as 0.58 is not uncommon. "def" = 0.50

**Contraction Pressure  
Drop Coefficient**

Coefficient used to account for additional pressure losses due to contraction at the inlet of the filter. If quadratic pressure drop relationship appears at high flow rates, then it may be modeled by this attribute.

$$\Delta P_1 = \frac{\xi_{contraction} \rho_{inlet} U_{inlet}^2}{2}$$

where  $\xi_{contraction}$  is the contraction coefficient (unitless). Typical values may range from 0.3-2.5.

**Expansion Pressure  
Drop Coefficient**

Coefficient used to account for additional pressure losses due to expansion at the outlet of the filter. If quadratic pressure drop relationship appears at high flow rates, then it may be modeled by this attribute.

$$\Delta P_{expansion} = \frac{\xi_{expansion} \rho_{outlet} U_{outlet}^2}{2}$$

where  $\xi_{expansion}$  is the expansion coefficient (unitless). Typical values may range from 0.3-2.5.

**Forchheimer Constant  
for Substrate Wall (1/m)**

In addition to the linear Darcy pressure drop, the quadratic pressure drop relationship with flow rate can also be modeled by including the Forchheimer term. This term can often be set to "ign", because the quadratic pressure drop behavior for modern wall-flow filters is only observed at high flow rates, typically higher than the flow rate of the fastest operating speed of the engine.

The overall pressure drop through the wall is calculated via:





$$\Delta P_{\text{Wall}} = \Delta P_{\text{Darcy}} + \Delta P_{\text{Forchheimer}}$$

Where:

$$\Delta P_{\text{Forchheimer}} = \beta_{\text{wall}} \rho_w U^2$$

Where  $\beta_{\text{wall}}$  is called the Forchheimer constant (Unit: 1/m). Typical values range from 0 to 5E9 1/m. If quadratic pressure drop vs. flow rate is observed for the wall-flow filter, "def" can be entered for this attribute to have the Forchheimer Constant calculated from the following formula.

$$\beta_{\text{wall}} = \frac{0.134}{\varepsilon^{1.5} k^{0.5}}$$

Where  $\varepsilon = \text{porosity}$  and  $k = \text{permeability}$ . The value of 0.134 is for smooth sphere packed beds. The porosity is the attribute **Filter Porosity** for the clean filter and the slab porosity calculated by the solver for the loaded filter. The permeability is the **Clean Filter Wall Permeability** for the clean filter and the **Loaded Filter Wall Permeability** for a loaded filter. The formula comes from SAE paper 2003-01-0846.

#### Pressure Drop Multiplier (Obsolete)

Multiplier to the total pressure drop across the filter. It is recommended that this attribute always be set to default ("def"=1). This attribute can be used to adjust pressure drop to match experimental data for QS solver only, however, it is **NOT recommended** and should be a last resort. All other input geometry should be double-checked first, and **Clean Filter Wall Permeability** above should be calibrated within acceptable range, before this attribute is adjusted. **Adjusting this attribute creates an unphysical solution (momentum not conserved) when the flow solver is set to explicit or implicit. This attribute will be phased out in a future version of the software.**

#### Improved Mixture Properties Calculation

Turning this checkbox on activates an improved method for calculation of the mixture conductivity and viscosity based on composition and temperature. This checkbox evolves to off to preserve old results; however we recommend that the checkbox be turned on for all new model calibrations for best results.

Previously the mixture properties were only a function of temperature per the Bissett 1984 journal paper. The improved method uses the same function as used in the pipes and flowsplits. The result of this is a significant change in viscosity, which directly affects the pressure drop across the substrate wall.

#### Improved Gas Density Calculation

Turning this checkbox on activates an improved method for calculation of the gas density through the soot cake and substrate wall using an average pressure from inlet channel to outlet channel rather than using local pressure. This checkbox evolves to off to preserve old results; however we recommend that the checkbox be turned on for all new model calibrations for best results. This improvement has the potential





to change the pressure drop by 10% during soot regeneration when species diffusion is activated.

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the wall-flow filter. In most cases, the **Calculated Wall Temperature** option will give the most realistic results.

#### ☉ Imposed Wall Temperature

Imposed wall temperature. Entering "def" for this attribute means the imposed wall temperature will be set equal to the initial gas temperature specified in the **Initial State Object** in the Main folder

#### ☉ Calculated Wall Temperature

Solve for the substrate and outer layer wall temperatures, and the resulting heat transfer rate to the fluid, accounting for wall material properties and external boundary conditions surrounding the filter.

#### (☉) Initial Wall Temperature

Temperature of the substrate and outer layer walls at the first time step of the simulation. Entering "def" for this attribute means the initial wall temperature will be set equal to the initial gas temperature specified in the **Initial State Object** in the Main folder

#### (☉) Substrate Thermal Properties Object

Name of the **'MaterialThermalProp'** reference object defining the material properties of the substrate for axial conductivity, density, and specific heat. The conductivity and density should include the porosity of the substrate, meaning  $(1-\text{porosity}) \times (\text{bulk solid property})$ . Predefined reference objects for substrate materials Cordierite and SiliconCarbide are available in the template library.

#### (☉) Outer Wall Layers Thermal Properties Object

Name of a **'WallThermalProperty'** object defining the emissivity, thickness, and material thermal properties of each outer layer. For example metal canning, insulation, air gap, heat shield, etc. Outer Layer 1 is in direct contact with the cylindrical surface of the substrate. This may be set to "ign" to ignore the thermal mass and resistance to external heat transfer of outer layers.

If outer layer 1 is using a **'MaterialThermalProp'** object with **Opacity** set to transparent, the radiation heat transfer for layer 1 will be ignored.

Direct convective heat loss from the substrate to the ambient can be modeled by setting this attribute to "ign" and defining a **'WallThermalBoundary'** object for the attribute **External Boundary Conditions Object** below with the **External Convection Coefficient** or **Free Convection Model** attributes defined in the object.

#### (☉) External Boundary Conditions Object

Name of a **'WallThermalBoundary'** object defining the external boundary conditions for calculating heat transfer to the ambient by convection and/or heat transfer to a black body by radiation.

Direct convective heat loss from the substrate to the ambient can be modeled by defining this object, and setting the **Outer Wall Layers Thermal Properties Object** to "ign" in the attribute above.

#### (☉) Wall Temperature

This flag determines what wall temperature solver setting is used:





## Solver Setting

- **same\_as\_thermal\_control:** This is the recommended setting for new models, and it causes the filter to use the global wall temperature solver flag under **Setup\Run Setup\ThermalControl\Thermal Solver**. The main purpose of this is to allow the global flag to be set to steady to activate a steady-state thermal solution for the filter in an engine+filter application model to allow it to converge to steady-state in less time. However, if there is soot loading or reactions activated in the filter, the local wall temperature solution will automatically revert back to transient. If performing an off/steady simulation, but the filter wall temperature is being solved transiently, then a temperature convergence RLT may need to be set under Run Setup -> ConvergenceRLT. This option is not supported for 0D (Number of Axial Intervals=1), 2D, and 3D.
- **transient:** Solves the wall temperature in the filter transiently using the thermal mass. All old models will evolve to this setting to preserve old results, because historically the filter wall temperature was always solved transiently, and ignored the global setting under **Setup\Run Setup\ThermalControl\Thermal Solver**.

## Additional Thermal Options

Specify optional thermal characteristics of the wall-flow filter.

### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the substrate will be scaled by this factor, but it only applies when the **Regeneration Object** is a 1D **'RegenReactions'** object. This attribute should typically be set to "def". ("def"=1.0)

### Heat Input Rate

The rate of heat input to the substrate. This attribute can be used when ignition energy is provided by an electric heater. The total power supplied will be uniformly distributed to the filter substrate. It should be set to "ign" if no electrical energy is supplied. This attribute may also be used for calibrating 1D heat loss, by making the heat input rate negative.

When this attribute is defined the **Wall Temperature Solver Object** attribute must be defined. The Heat Input Rate may be specified with reference objects or actuated via an ActuatorConn for use in a control system, and the actuated value will overwrite the attribute value.

### Thermal Solution Method

This attribute determines which method is used in the thermal wall solution of the filter. The following options are available:

- **rk-integration:** this method uses direct integration via the "Runge-Kutta" method to determine the wall temperature.
- **NTU:** this method uses the "Number of Transfer Units" to determine the wall temperature.

The NTU method has been found to be a more stable, more robust, and faster solution method than the rk-integration method. For most applications the NTU method is highly recommended. NTU method is also required if the Ash Model is turned ON in the **'Filtration'** object. At high flow rates and steady inlet conditions the two methods should produce approximately the same results. The rk-integration method may







be necessary if the incoming mass flow rate, temperature, or species concentration is changing very rapidly (step change). The rk-integration method may be more accurate at low flow rates when a fine axial discretization is used.

## Modeling Options

### Filtration Object

Name of the **'Filtration'** reference object that describes the particulate matter filtration and loaded filter pressure drop model. This attribute can be set to "ign" if a clean filter is modeled. In such case, pressure drop is evaluated strictly from information provided in the Pressure Drop folder.

### Regeneration Kinetics Flag

This flag determines if regeneration and catalytic reactions are solved:

- **built-in:** use the predefined kinetic reaction mechanism present in the **'DPFReg0D'** or **'RegenReactions'** called from the **Regeneration Object** attribute below.
- **external:** use a user-defined mechanism using either the **'GlobalReactions'** or **'SurfaceReactions'** template. If this option is selected, the user must connect one of these reaction parts to the **'ParticulateFilter'** via a **'ChemConn'** connection and point to the reaction part in the **'RegenReactions'** object. If "area" rate is selected in the global reactions part, the specific reaction area is  $1 \text{ m}^2/\text{m}^3$ .
- **off:** ignore the **Regeneration Object** attribute below and thus don't model any reactions. By setting this attribute to "OFF" the user can simulate 1D clean filter pressure drop and soot loading pressure drop in conjunction with the **'Filtration'** reference object without needing to define the regeneration reactions.

### Regeneration Object

Name of the **'DPFReg0D'** or **'RegenReactions'** reference object describing the particulate matter oxidation and filter regeneration model. This attribute can be set to "ign" if a clean filter is studied or only particulate matter filtration is modeled. To model filter regeneration, a **'Filtration'** object must be provided. That means that filtration object cannot be set to "ign" when simulating filter regeneration.

### 2D/3D Object

Name of the **'ExhaustATDevice2D'** or **'ExhaustATDevice3D'** reference object describing the particulate matter oxidation and filter regeneration model. This attribute can be set to "ign". To model either 2D or 3D, both the **'Filtration'** and the **'RegenReactions'** reference objects must be defined.

## Solver Options

The attributes in this folder are used when Regeneration Kinetics Flag = "built-in". The RK-related attributes are also used if the Thermal Solution Method is set to "RK".

### Solver Settings for Regeneration

These attributes are associated with regeneration kinetics solution.

### ODE/DAE Solver Selection

There are two available ODE/DAE solver methods:







- **Adaptive RK** indicates "Adaptive Runge-Kutta" ODE method, which is more efficient than the **BDF** method and is most suitable for non-stiff ODE system (A reaction system is called stiff when it includes both very fast and very slow dynamics, that is, some reactions are much faster than the others).
- **BDF** indicates "Backward Differential Formulations" DAE method, which is more robust than the **Adaptive RK** method and is most suitable for handling stiff ODE system.

The **BDF** selection is recommended for all soot regeneration applications.

**RK Overall Error Tolerance**

Tolerance used to control the overall error in Runge-Kutta method. ("def" = 1.0E-4)

**RK Multiplier for First Step**

Multiplier to master time step size for determining the first Runge-Kutta step size. ("def" = 1.0)

**RK Step Size Adjusting Factor**

Adjusting factor controlling the degree of relaxation. ("def" = 5.0)

**RK Multiplier for Minimum Step Size**

Multiplier to the master time step size for determining the minimum step size RK can use. ("def" = 1.0E-10)

**BDF Absolute Tolerance**

Absolute error tolerance in BDF method. ("def" = 1.0E-12)

**BDF Relative Tolerance**

Relative error tolerance in BDF method. ("def" = 1.0E-3)

**Corrected Volume Rate (External Reactions)**

Turning this checkbox on activates a correction to the total volume rate and site (turnover number) rate forms when external reactions are attached to the '[ParticulateFilter](#)'. This checkbox evolves to off to preserve old results; however, we recommend that the checkbox be turned on for all new model calibrations for best results.

**Corrected Soot Concentration (External Reactions)**

This attribute determines the definition of the soot concentration term (such as {C}) that may be found in an external reaction part, such as '[GlobalReactions](#)' or '[SurfaceReactions](#)', being used for regeneration reactions. When this checkbox is activated, soot concentration terms are interpreted as the local mass of soot in grams per unit volume (subvolume) of reactor. This definition holds for reactions in both the soot cake layer and the wall, and activating this checkbox is recommended.

Deactivating this checkbox causes soot concentration terms to be evaluated differently depending on whether the reaction is occurring in the soot cake layer or the wall. For soot reactions in the soot cake layer, the soot concentration term is interpreted as the local mass of soot per unit volume of soot cake. The user is reminded that this soot cake density is typically a constant value. For soot reactions in the wall, the soot concentration term is interpreted as the local mass of soot per unit wall volume. Deactivating this checkbox preserves model behavior from V7.5 and earlier.

These attributes are associated with the wall-flow filter flow solution.

**Flow Numerical Control**





### Time Step for Explicit Flow Solver

Simulation time step for Explicit Solver only. Numerical value or "def" may be specified. If "def" is specified then one of the following will occur:

1. If connected to a pipe or flowsplit the minimum timestep taken from the flow simulation will be applied.
2. When 'ParticulateFilter' is connected to 'End\*' (i.e. 'EndFlowInlet' and 'EndEnvironment') type components, the code will calculate a computationally efficient timestep based on the retention time which will ensure numerical stability.

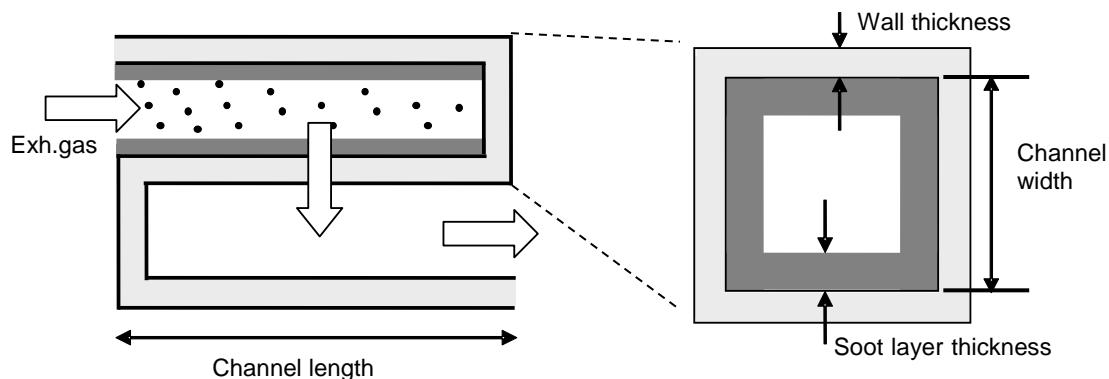
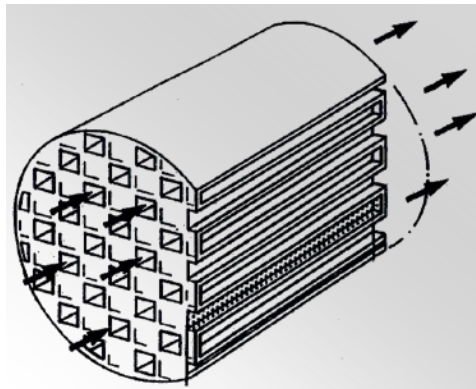
For most applications, a value of "def" should be specified.

### Maximum Matrix Inversion Iterations

Matrix inversion is the default technique solving the two point boundary problem for flow solution. This attribute indicates the maximum number of iterations that the solver will take on each time step. ("def" = 10)

### Matrix Inversion Convergence Criteria

The tolerance, when the matrix inversion technique is utilized, to which the relative error of a flow variable at two consequent iteration steps should be reduced at each time step. ("def" =  $1.0e-4$ )



**Schematic of a Wall-Flow Particulate Filter (top) and Channel Configuration (bottom)**





## GEMPerfAll - Perforation over Entire Component

This template is used to model perforations covering an entire section of a component. Perforations consist of many small diameter holes covering the entire baffle area.

*\* It is important to note that perforate sections are intended to be used to model relatively small holes when there are many of them. When there are only a few medium to large diameter holes, they should be modeled individually using the 'GEMOrifice', 'GEMOrificeBaffle', 'GEMOrificePipe', or 'GEMOrificePipeBend' templates. If a perforate section is used in these situations, it is likely GEM will give an error because the number of holes is less than the number of connections being made, resulting in less than 1 hole per orifice, which is not allowed. The following 3 ideas may prevent this situation.*

- Perforate does not have enough holes. For sections with only a few holes individual orifices should be added to the baffle rather than a perforated section.
- Perforate hole(s) lying on the boundary of multiple shell cubes (flowsplits). Increasing the number of holes will likely fix this problem. In some situations, the position of the perforate section can be modified to avoid a shell cube boundary.
- Many perforate sections can be combined into a single perforate section.

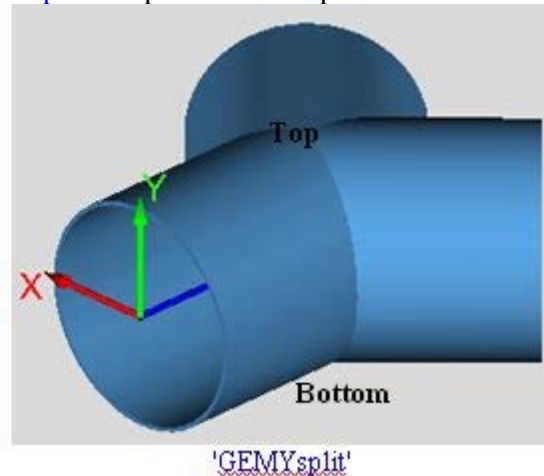
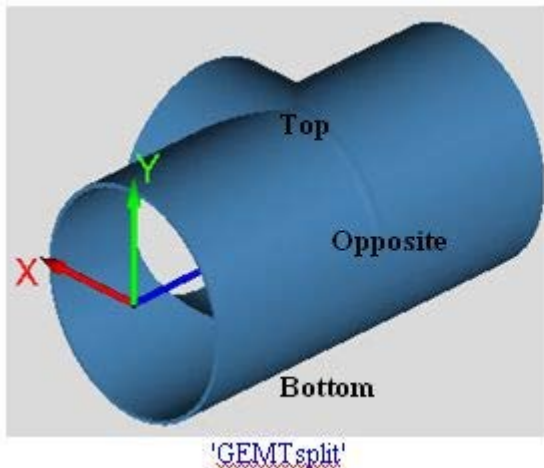
## Geometry

<b>Hole Diameter</b>	Hole diameter.
<b>Number of Identical Holes</b>	Specifies the total number of holes to include in the perforated section. "def" may be specified if the <b>Porosity</b> is specified below. In this case GEM3D will calculate the number of holes during discretization based on the total area and the <b>Porosity</b> .
<b>Porosity</b>	Specifies the porosity of the perforation. "def" may be specified if the <b>Number of Identical Holes</b> is specified above. In this case GEM3D can calculate the porosity (see the <a href="#">Show Porosity</a> operation) based on the total area and the <b>Number of Identical Holes</b> .
<b>Section to Perforate</b>	<p>Specifies the section of the parent component to perforate. For the 'GEMTsplit' or 'GEMYsplit' components refer to the drawing below. Choices include:</p> <ul style="list-style-type: none"><li>• <b>Entire</b> indicates that the entire component will be perforated. This is the only available choice for perforating baffles (and thus cannot be edited). This is the default choice for other components.</li><li>• <b>Top</b> indicates that the top section of a 'GEMTsplit' or 'GEMYsplit' component will be perforated.</li><li>• <b>Bottom</b> indicates that the bottom section of a 'GEMTsplit' or 'GEMYsplit' component will be perforated.</li><li>• <b>Opposite</b> indicates that the side opposite the tee port of a 'GEMTsplit' component will be perforated.</li><li>• <b>Top and Bottom</b> indicates that the top and bottom sections of a 'GEMTsplit' component will be perforated.</li><li>• <b>Top and Opposite</b> indicates that the top and opposite sections of a 'GEMTsplit' component will be perforated.</li></ul>





- **Bottom and Opposite** indicates that the bottom and opposite sections of a 'GEMTsplit' component will be perforated.



## Main

### Forward Discharge Coefficient

Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Reverse Discharge Coefficient

Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

## Options

### Forward End Correction (Length/Diameter)

End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Reverse End Correction (Length/Diameter)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume





## Heat Conduction "Flange"

of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

## Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one '[OrificeConn](#)' connection in a string of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def". The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery** This option enables the pressure recovery in orifice connections as described in the Flow Theory Manual. This option should always be used for sudden contractions and expansions and the discharge coefficients should be set to "def" to be automatically calculated. If this option is used and discharge coefficients are specified, the resulting pressure drop/flow rate may not be equal to Bernoulli's Equation due to the pressure recovery calculated. This was the setting that was used in all previous versions.
- **NoRecovery** This option disables the pressure recovery in the orifice. In other words the pressure in the vena contracta (throat of the orifice) and the downstream will be equal.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option





is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def" and the **Laminar Face Friction Multiplier** should be set to 0.

With this choice, the pressure drop/flow rate under steady state conditions will match Bernoulli's Equation, assuming friction and heat transfer are ignored. (Note that GT-SUITE never uses the Bernoulli equation directly. It always solves the momentum equation.)

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is unphysical and therefore not allowed.





## GEMPerfCS - Perforated Section of a Baffle

This template is used to model a perforated section of a baffle. Perforations consist of many small diameter holes covering a section of the baffle. This can be done using the template attributes at the top of the window and the 2D graphical interface at the bottom of the window. This editor is used to choose and position a cross section on the baffle to be used as a perforate section. The graphical window will automatically contain the shape of the baffle. This allows the perforated cross section to be placed anywhere on the baffle by placing it in the desired location in the graphical window. An existing or new cross section can be added to the graphical window to use it as the perforated section shape. This section can then be placed interactively in the graphical window to position the perforation or the template location attributes may be used. The interface in the graphical window is essentially the same as the [Cross Section Editor](#), however, with some restricted capabilities.

*\* It is important to note that perforate sections are intended to be used to model relatively small holes and when there are many of them. When there are only a few medium to large diameter holes, they should be modeled individually using the 'GEMOrifice', 'GEMOrificeBaffle', 'GEMOrificePipe', or 'GEMOrificePipeBend' templates. If a perforate section is used in these situations, it is likely GEM will give an error because the number of holes is less than the number of connections being made, resulting in less than 1 hole per orifice, which is not allowed. The following 3 ideas may prevent this situation.*

- Perforate does not have enough holes. For sections with only a few holes individual orifices should be added to the baffle rather than a perforated section.
- Perforate hole(s) lying on the boundary of multiple shell cubes (flowsplits). Increasing the number of holes will likely fix this problem. In some situations, the position of the perforate section can be modified to avoid a shell cube boundary.
- Many perforate sections can be combined into a single perforate section.

## Geometry

<b>Cross Section</b>	Name of the cross section object describing the shape of the perforated section. Any of the 4 standard cross section shapes ( <a href="#">circle</a> , <a href="#">ellipse</a> , <a href="#">rectangle</a> , <a href="#">rounded rectangle</a> ) may be used. Alternatively the <b>Add Cross Section</b> option from the graphical window interface below the template window may be used (see commands after this table).
<b>Location X</b>	Specifies the X location on the baffle of the cross section's centroid. This location is relative to the <a href="#">local components axis</a> .
<b>Location Y</b>	Specifies the Y location on the baffle of the cross section's centroid. This location is relative to the <a href="#">local components axis</a> .
<b>Hole Diameter</b>	Hole diameter.
<b>Number of Identical Holes</b>	Specifies the total number of holes to include in the perforated section. "def" may be specified if the <b>Porosity</b> is specified below. In this case GEM3D will calculate the number of holes during discretization based on the total area and the <b>Porosity</b> .
<b>Porosity</b>	Specifies the porosity of the perforation. "def" may be specified if the <b>Number of Identical Holes</b> is specified above. In this case GEM3D can calculate the porosity (see the <a href="#">Show Porosity</a> operation) based on the total area and the <b>Number of Identical Holes</b> .







---

## Main

---

### Forward Discharge Coefficient

Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Reverse Discharge Coefficient

Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

---

## Options

---

### Forward End Correction (Length/Diameter)

End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Reverse End Correction (Length/Diameter)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Heat Conduction "Flange"

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

### Initial Mass Flow Rate



Initial mass flow rate of the fluid at the connection. This attribute is





typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one 'OrificeConn' connection in a strong of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def" The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery** This option enables the pressure recovery in orifice connections as described in the Flow Theory Manual. This option should always be used for sudden contractions and expansions and the discharge coefficients should be set to "def" to be automatically calculated. If this option is used and discharge coefficients are specified, the resulting pressure drop/flow rate may not be equal to Bernoulli's Equation due to the pressure recovery calculated. This was the setting that was used in all previous versions.

- **NoRecovery** This option disables the pressure recovery in the orifice. In other words the pressure in the vena contracta (throat of the orifice) and the downstream will be equal.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def" and the **Laminar Face Friction Multiplier** should be set to 0.

With this choice, the pressure drop/flow rate under steady state conditions will match Bernoulli's Equation, assuming friction and heat transfer are ignored. (Note that GT-SUITE never uses the Bernoulli equation directly. It always solves the momentum equation.)

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is unphysical and therefore not allowed.

The commands used in the graphical window to place the perforated cross sections are available on the toolbar above the graphical window. They are described below.





**Undo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row.



**Redo:** Undoes the last operation. This operation can be used sequentially to undo multiple operations that were done in a row.



**Toggle Snap to Grid:** Toggles the snap to grid function. With this enabled, all locations will snap to the nearest grid point.



**Toggle Grid:** Toggles the display of the grid.



**Zoom In:** Zooms in on the cross section. [Keyboard hotkey = NumPad +]



**Zoom Out:** Zooms out from the cross section. [Keyboard hotkey = NumPad -]



**Zoom 1:1:** Restores the zoom to the default level (1:1).



**Fit to Screen:** Centers the cross section and zooms to the appropriate level to fit the entire cross section to the editor window.



**Center Canvas:** Centers the canvas on the origin without changing the zoom level.

**Unit:** Drop down menu specifying the units for the cross sections. Choices include m, cm, mm, in, ft, micron, km, mi, and dm.



**Rotate:** Rotates the cross section used as the perforate section in the plane of the parent baffle.



**Add Cross Section:** Adds an existing cross section or allows the creation of a new cross section to be used as the perforate section. This will open the cross section manager window that will allow selection of any existing cross sections or the creation of new ones.



**Delete Cross Section:** Deletes the current cross section from the graphical window only. This means that the perforate section is deleted, not that the actual cross section is deleted. The actual cross section will still be available in the model.



**Configure Ruler:** Allows the ruler spacing to be manually specified.





## GEMPerfPoint - Perforated Section on the surface of a Shell

This template is used to model a perforated section on the surface of a shell. Perforations consist of many small diameter holes covering a section of the shell surface. The perforated section must have a circular shape and be on the surface of the shell.

*\* It is important to note that perforate sections are intended to be used to model relatively small holes and when there are many of them. When there are only a few medium to large diameter holes, they should be modeled individually using the 'GEMOrifice', 'GEMOrificeBaffle', 'GEMOrificePipe', or 'GEMOrificePipeBend' templates. If a perforate section is used in these situations, it is likely GEM will give an error because the number of holes is less than the number of connections being made, resulting in less than 1 hole per orifice, which is not allowed. The following 3 ideas may prevent this situation.*

- *Perforate does not have enough holes. For sections with only a few holes individual orifices should be added to the baffle rather than a perforated section.*
- *Perforate hole(s) lying on the boundary of multiple shell cubes (flowsplits). Increasing the number of holes will likely fix this problem. In some situations, the position of the perforate section can be modified to avoid a shell cube boundary.*
- *Many perforate sections can be combined into a single perforate section.*

### Geometry

<b>Location X (Local)</b>	Specifies the local X location on the shell of the perforated section's center. This location is relative to the shell's <a href="#">local components axis</a> .
<b>Location Y (Local)</b>	Specifies the local Y location on the shell of the perforated section's center. This location is relative to the shell's <a href="#">local components axis</a> .
<b>Location Z (Local)</b>	Specifies the local Z location on the shell of the perforated section's center. This location is relative to the shell's <a href="#">local components axis</a> .
<b>Direction X (Local)</b>	Specifies the X component of the vector describing the direction that the perforate section will be created. Only a unit vector is needed to describe the direction, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y (Local)</b>	Specifies the Y component of the vector describing the direction that the perforate section will be created. Only a unit vector is needed to describe the direction, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z (Local)</b>	Specifies the Z component of the vector describing the direction that the perforate section will be created. Only a unit vector is needed to describe the direction, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Diameter of Perforate Section</b>	Specifies the diameter of the section to be perforated.
<b>Hole Diameter</b>	Specifies the diameter of an individual hole.
<b>Number of Identical Holes</b>	Specifies the total number of holes to include in the perforated section. "def" may be specified if the <b>Porosity</b> is specified below. In this case





GEM3D will calculate the number of holes during discretization based on the total area and the **Porosity**.

## Porosity

Specifies the porosity of the perforation. "def" may be specified if the **Number of Identical Holes** is specified above. In this case GEM3D can calculate the porosity (see the [Show Porosity](#) operation) based on the total area and the **Number of Identical Holes**.

## Main

### Forward Discharge Coefficient

Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Reverse Discharge Coefficient

Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

## Options

### Forward End Correction (Length/Diameter)

End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Reverse End Correction (Length/Diameter)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

### Heat Conduction "Flange"

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is





added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

## Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one 'OrificeConn' connection in a string of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def". The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery** This option enables the pressure recovery in orifice connections as described in the Flow Theory Manual. This option should always be used for sudden contractions and expansions and the discharge coefficients should be set to "def" to be automatically calculated. If this option is used and discharge coefficients are specified, the resulting pressure drop/flow rate may not be equal to Bernoulli's Equation due to the pressure recovery calculated. This was the setting that was used in all previous versions.
- **NoRecovery** This option disables the pressure recovery in the orifice. In other words the pressure in the vena contracta (throat of the orifice) and the downstream will be equal.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def" and the **Laminar Face Friction Multiplier** should be set to 0.

With this choice, the pressure drop/flow rate under steady state conditions will match Bernoulli's Equation, assuming friction and heat transfer are ignored. (Note that GT-SUITE never uses the Bernoulli equation directly. It always solves the momentum equation.)

This choice should not be used to model a pure sudden expansion,





since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is unphysical and therefore not allowed.





## GEMPerfRef - Perforated Section of a Straight Pipe, Shell, or Flowsplit

This template is used to model perforated sections of pipes, shells, and flowsplits. Perforations consist of many small diameter holes covering a section of a pipe, shell, and/or flowsplit. The perforated section will be given as a length of the component to perforate. This section can cover all the way around the pipe or only partially around the pipe.

- For pipes, the perforated section may be axially discretized if the length of the perforate section is longer than the pipe discretization length specified during the discretization process.
- For flowsplits, since they are only discretized as a single flowsplit in the model file, the perforated section will not be axially discretized.
- For shells, the perforate section may be axially discretized depending on the length of the perforate section and the discretization of the shell itself.

*\* It is important to note that perforate sections are intended to be used to model relatively small holes and when there are many of them. When there are only a few medium to large diameter holes, they should be modeled individually using the 'GEMOrifice', 'GEMOrificeBaffle', 'GEMOrificePipe', or 'GEMOrificePipeBend' templates. If a perforate section is used in these situations, it is likely GEM will give an error because the number of holes is less than the number of connections being made, resulting in less than 1 hole per orifice, which is not allowed. The following 3 ideas may prevent this situation.*

- *Perforate does not have enough holes. For sections with only a few holes individual orifices should be added to the baffle rather than a perforated section.*
- *Perforate hole(s) lying on the boundary of multiple shell cubes (flowsplits). Increasing the number of holes will likely fix this problem. In some situations, the position of the perforate section can be modified to avoid a shell cube boundary.*
- *Many perforate sections can be combined into a single perforate section.*

## Geometry

<b>Hole Diameter</b>	Hole diameter.
<b>Number of Identical Holes</b>	Specifies the total number of holes to include in the perforated section. "def" may be specified if the <b>Porosity</b> is specified below. In this case GEM3D will calculate the number of holes during discretization based on the total area and the <b>Porosity</b> .
<b>Porosity</b>	Specifies the porosity of the perforation. "def" may be specified if the <b>Number of Identical Holes</b> is specified above. In this case GEM3D can calculate the porosity (see the <a href="#">Show Porosity</a> operation) based on the total area and the <b>Number of Identical Holes</b> .
<b>Distance from Reference</b>	Specifies the distance along the component from the reference where the perforated section will begin. "ign" can be specified when <b>Reference</b> is set to <b>Centered</b> .
<b>Reference</b>	Specifies the reference location of the component in which to position the perforated section. Choices include: <ul style="list-style-type: none"> <li>• <b>From start</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the perforated section from the start of the</li> </ul>





pipe (first cross section).

- **From end** indicates that the **Distance from Reference** value above specifies the position of the perforated section from the end of the pipe (last cross section).
- **Centered** indicates that the perforate section will be centered on the pipe. The **Distance from Reference** value above should be set to "ign".

#### Length of Perforate Section

Length of the perforated section.

#### Angle of Coverage

Specifies the portion of the component to perforate as an angle around the component. A value of 360 degrees would perforate all the way around the component. A value of 180 would only perforate half the way around the component.

#### Start of Coverage

Specifies the position around the component to begin the perforations as an angle. A value of 0 degrees would begin the perforation aligned with the major axis of the first cross section.

### Main

#### Forward Discharge Coefficient

Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

#### Reverse Discharge Coefficient

Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

### Options

#### Forward End Correction (Length/Diameter)

End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

#### Reverse End Correction (Length/Diameter)

End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent







components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does not recommend a user to enter another value under any circumstances. ("def" = 0.355)

## Heat Conduction "Flange"

Name of the '[WallConductionConn](#)' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

## Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one '[OrificeConn](#)' connection in a string of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def". The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery** This option enables the pressure recovery in orifice connections as described in the Flow Theory Manual. This option should always be used for sudden contractions and expansions and the discharge coefficients should be set to "def" to be automatically calculated. If this option is used and discharge coefficients are specified, the resulting pressure drop/flow rate may not be equal to Bernoulli's Equation due to the pressure recovery calculated. This was the setting that was used in all previous versions.
- **NoRecovery** This option disables the pressure recovery in the orifice. In other words the pressure in the vena contracta (throat of the orifice) and the downstream will be equal.





This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def" and the **Laminar Face Friction Multiplier** should be set to 0.

With this choice, the pressure drop/flow rate under steady state conditions will match Bernoulli's Equation, assuming friction and heat transfer are ignored. (Note that GT-SUITE never uses the Bernoulli equation directly. It always solves the momentum equation.)

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is unphysical and therefore not allowed.





## GEMPerfRefBend - Perforated Section of a Bent Pipe

This template is used to model perforated sections of bent pipes. Perforations consist of many small diameter holes covering a section of a bent pipe. The perforated section will be given as a length of the pipe to perforate. This section can cover all the way around the pipe or only partially around the pipe.

*\* It is important to note that perforate sections are intended to be used to model relatively small holes and when there are many of them. When there are only a few medium to large diameter holes, they should be modeled individually using the 'GEMOrifice', 'GEMOrificeBaffle', 'GEMOrificePipe', or 'GEMOrificePipeBend' templates. If a perforate section is used in these situations, it is likely GEM will give an error because the number of holes is less than the number of connections being made, resulting in less than 1 hole per orifice, which is not allowed. The following 3 ideas may prevent this situation.*

- *Perforate does not have enough holes. For sections with only a few holes individual orifices should be added to the baffle rather than a perforated section.*
- *Perforate hole(s) lying on the boundary of multiple shell cubes (flowsplits). Increasing the number of holes will likely fix this problem. In some situations, the position of the perforate section can be modified to avoid a shell cube boundary.*
- *Many perforate sections can be combined into a single perforate section.*

## Geometry

<b>Hole Diameter</b>	Hole diameter.
<b>Number of Identical Holes</b>	Specifies the total number of holes to include in the perforated section. "def" may be specified if the <b>Porosity</b> is specified below. In this case GEM3D will calculate the number of holes during discretization based on the total area and the <b>Porosity</b> .
<b>Porosity</b>	Specifies the porosity of the perforation. "def" may be specified if the <b>Number of Identical Holes</b> is specified above. In this case GEM3D can calculate the porosity (see the <a href="#">Show Porosity</a> operation) based on the total area and the <b>Number of Identical Holes</b> .
<b>Distance from Reference</b>	Specifies the distance along the pipe from the reference where the perforated section will begin. "ign" can be specified when <b>Reference</b> is set to <b>Centered</b> .
<b>Reference</b>	Specifies the reference location of the component in which to position the perforated section. Choices include: <ul style="list-style-type: none"> <li>• <b>From start</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the perforated section from the start of the pipe (first cross section).</li> <li>• <b>From end</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the perforated section from the end of the pipe (last cross section).</li> <li>• <b>Centered</b> indicates that the perforated section will be centered on the pipe. The <b>Distance from Reference</b> value above should be set to "ign".</li> </ul>
<b>Length of Perforate</b>	Length of the perforated section.





## Section

<b>Angle of Coverage</b>	Specifies the portion of the pipe to perforate as an angle around the pipe. A value of 360 degrees would perforate all the way around the pipe. A value of 180 would only perforate half the way around the pipe.
<b>Start of Coverage</b>	Specifies the position around the pipe to begin the perforations as an angle. A value of 0 degrees would begin the perforation aligned with the major axis of the first cross section.

## Main

<b>Forward Discharge Coefficient</b>	Discharge coefficient in the direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.
<b>Reverse Discharge Coefficient</b>	Discharge coefficient in the reverse direction of the linking arrows or the name of a dependency reference object. If "def" is entered, the coefficient will be automatically calculated using the geometry of the mating flow components and the orifice diameter, assuming that all transitions are sharp-edged.

## Options

<b>Forward End Correction (Length/Diameter)</b>	End correction for the orifice in the forward direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does <u>not</u> recommend a user to enter another value under <u>any</u> circumstances. ("def" = 0.355)
<b>Reverse End Correction (Length/Diameter)</b>	End correction for the orifice in the reverse direction. An end correction adds a "virtual mass" at the orifice when there is a large area change. This "virtual mass" is involved in the momentum equation at the orifice, and effectively changes the distance that a wave in the adjacent components will travel before reflecting at the area change. The end correction is "virtual" and does not change the pipe length or the volume of the adjoining component. The end correction is entered as the "virtual mass length" as a fraction of the orifice diameter. GT recommends very strongly that users leave this value as "def" and does <u>not</u> recommend a user to enter another value under <u>any</u> circumstances. ("def" = 0.355)
<b>Heat Conduction "Flange"</b>	Name of the 'WallConductionConn' reference object describing the thermal mass of a flange at the orifice. This object also enables heat conduction across this connection between two adjacent pipes and/or





flowsplits. "def" causes the adjacent components to be fused together for conduction heat transfer. In this case, no additional thermal mass is added to the flange. This attribute can be set to "ign" and this will suppress heat conduction across this connection and set the flange thermal mass to zero.

The flange option is not allowed for flow connections placed between a conventional flow component ('Pipe\*', 'FlowSplit\*', etc.) and a flow component that models its wall with a thermal component.

## Initial Mass Flow Rate

Initial mass flow rate of the fluid at the connection. This attribute is typically only used for liquids modeling where discontinuities in the initial mass flow rates can cause large pressure spikes due to the incompressibility of the liquid. This attribute should typically be set to "def" for gases.

The values specified in this attribute will smartly propagate through a flow system. For example, if the initial mass flow rate is set in one 'OrificeConn' connection in a string of pipes, that value will propagate to all adjacent orifices in which this attribute is set to "def". The value will also propagate through flowsplits with exactly 2 connections. For flowsplits with 3 or more branches, the propagation will stop at the flowsplit until it can be resolved. For an example of a 4 branch flowsplit, if three branches are initialized mass flow rates, the 4th branch will be initialized after the other 3 branches by summing the initial flow rates of the other three branches.

## Pressure Recovery Choice

This attribute controls the method of predicting the pressure loss across the connection :

- **PressureRecovery** This option enables the pressure recovery in orifice connections as described in the Flow Theory Manual. This option should always be used for sudden contractions and expansions and the discharge coefficients should be set to "def" to be automatically calculated. If this option is used and discharge coefficients are specified, the resulting pressure drop/flow rate may not be equal to Bernoulli's Equation due to the pressure recovery calculated. This was the setting that was used in all previous versions.
- **NoRecovery** This option disables the pressure recovery in the orifice. In other words the pressure in the vena contracta (throat of the orifice) and the downstream will be equal.

This option should be used when discharge coefficients calculated from flow test bench measurements are to be imposed. If this option is used, the Forward and Reverse Discharge Coefficients have to be specified and cannot be set to "def" and the **Laminar Face Friction Multiplier** should be set to 0.

With this choice, the pressure drop/flow rate under steady state conditions will match Bernoulli's Equation, assuming friction and heat transfer are ignored. (Note that GT-SUITE never uses the Bernoulli equation directly. It always solves the momentum





equation.)

This choice should not be used to model a pure sudden expansion, since the mass flow rate will be based on the smaller diameter. Achieving the ideal mass flow rate would therefore require discharge coefficients greater than 1, which is unphysical and therefore not allowed.





## GEMPipeXYZPoints - Pipe Formed Using X, Y, and Z Coordinates

This template is used to describe a pipe section that is specified using a series of points. The points are specified using a relative X, Y, and Z location for each. The pipe section is then formed by fitting a cross section to the specified points.

### Geometry

#### Bend Radius

Radius of the pipe's bend measured from the center of curvature to the centroid of the pipe's cross-sectional area. If the **Fitting Method=Spline** or the **Fitting Method=Spline-Modified**, then this attribute may be set to "def" and GEM3D will calculate the correct bend radius. If the **Fitting Method=Line-Arcs**, then this attribute cannot be set to "def" and must be given a value.

#### Fitting Method

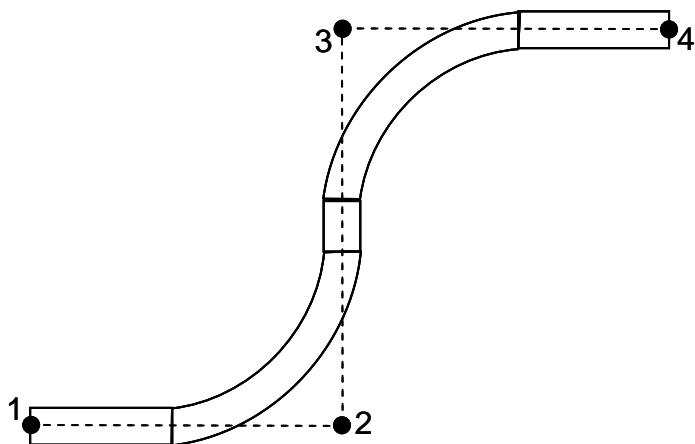
Specifies the fitting method to be used on the coordinates on the XYZPoints tab to create the pipe section. Choices include:

- **Spline-Modified** this is a modified spline method that does not have to pass through every specified point. This method forces the end cross sections to be normal to the direction specified by the last 2 points of the spline.
- **Line-Arcs** indicates the pipe will be formed from the specified coordinates on the XYZPoints tab using the Line-Arc method. This method requires the **Bend Radius** attribute above to be specified. This method assumes that a straight line connects between each point. The difference in angle between the lines gives the bend angle of the pipe at that point. The **Bend Radius** gives the bend radius to be used for the bent pipe section. The discretized result will be a combination of alternating straight and bent pipes in the GT-SUITE model. See the figure below to see a drawing of the Line-Arc method.

#### Wall Thickness

Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a ['WallThermalProperty'](#) reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the ['WallThermalProperty'](#) reference object.





**Line-Arc Fitting Method** - Only 4 points are needed to describe this section. The bent and straight sections can be calculated from the location of the points and the imposed **Bend Radius**.

### XYZPoints

*The attributes in this folder specify the X, Y, and Z coordinates of each point to be used to fit the pipe section. Each row specifies a single point.*

<b>Name</b>	Specifies the name of each point. This name is only used as a label in the component and will not affect the actual building or discretization of the component.
<b>X</b>	Specifies the local X value of the series of points. The relative location of the specific points will be used to determine the pipe's overall geometry. The pipe will then be located in the global coordinate system according to the values specified on the Location tab.
<b>Y</b>	Specifies the local Z value of the series of points. The relative location of the specific points will be used to determine the pipe's overall geometry. The pipe will then be located in the global coordinate system according to the values specified on the Location tab.
<b>Z</b>	Specifies the local Z value of the series of points. The relative location of the specific points will be used to determine the pipe's overall geometry. The pipe will then be located in the global coordinate system according to the values specified on the Location tab.

### Cross Sections

<b>Length</b>	Length of a given section from the inlet of the pipe. Each column corresponds to a pipe section, and this value represents the distance of the beginning of that pipe section from the inlet of the pipe. Column 1 must have a value of "0", representing the inlet of the pipe. The total length of the pipe is calculated based on the geometry on the <b>XYZPoints</b> tab. The length specified here is normalized. This means the length in the final column will represent the end of the pipe, and all other lengths are normalized to that length.
---------------	--







**Cross Section Name** Name of the cross section object(s) describing the shape to be used for the XYZ pipe. The XYZ pipe allows any of the 4 standard cross section shapes ([circle](#), [ellipse](#), [rectangle](#), [rounded rectangle](#)) as well as a [custom shape](#). If only 1 cross section is specified, then that cross section will be used at the beginning and end of the pipe. If 2 or more are specified, then the first one will be used at the beginning of the pipe and the last one will be used at the end. The rest will be placed according to the Length above. The resulting pipe will use a smooth transition to change cross sections along the length of the pipe.

## Location

**Location X** Specifies the absolute X location of the component's first point from the XYZPoints tab.

**Location Y** Specifies the absolute Y location of the component's first point from the XYZPoints tab.

**Location Z** Specifies the absolute Z location of the component's first point from the XYZPoints tab.

**Direction X** Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

**Direction Y** Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

**Direction Z** Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

## Visual

**Transparency Percent** Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

**Display Color** Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper





- Gold
- Grey
- Black

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Discretization Length

Discretization length to be used for pipes during the discretization process. This length does not need to be an even fraction of the entire pipe length; the code will adjust to divide the pipe appropriately. A value of "def" will use the discretization length found in the global discretization window. See [export gtm](#) for additional information.

#### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

#### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

#### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25





- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

## © Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

## Additional Geometry Options

### Pipe Elevation Change or 3D Acceleration Object

Specify optional geometric characteristics of the flow component.

This attribute defines either the elevation change of the pipe from the inlet to the outlet, or 3D acceleration resolved along the pipe axis via a '[PipeMotion3DBM](#)' reference object. If "def" is entered, then the elevation change or 3D acceleration object of the pipe will be automatically generated from the option selected in the **Gravity/Acceleration Options** field of [File>Options>Discretization - Contains the discretization preferences](#).

When the gravity option is used, a positive value means port 2 is at a higher elevation than port 1, and vice versa. The acceleration due to gravity is assumed to be  $9.80665 \text{ m/s}^2$ . When '[XYTable](#)' reference object is used, X is defined as pipe length (which is normalized) and Y is the elevation change.

This attribute cannot be used in parallel with the **Body Force Acceleration** attribute under the Options folder.

## No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

## © Imposed Wall



Impose a temperature which represents the temperature of the wall



## Temperature

surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### ☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

### ☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

### ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses). This option should be used when this Pipe object will be an intake or exhaust port for a cylinder. This option should be used when this Pipe object will be an intake or exhaust port for a cylinder.

### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement





	Factor")
<b>Heat Input Rate</b>	The rate of heat input to the fluid or the name of a dependency reference object.
<b>Thermocouple Object</b>	Name of a <a href="#">'Thermocouple'</a> reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe* or FlowSplit* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.
<b>☉ Heat Transfer Correlation (Colburn)</b>	Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.
<b>☉ User Defined Heat Transfer Model</b>	Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to defined the name of the <a href="#">'UserModel'</a> object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the <b>Heat Transfer Multiplier</b> value.
<b>☉ Heat Transfer Coefficient</b>	Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.
<b>Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)</b>	<p>Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to <a href="#">'FluidRefrigerant'</a> Circuits, which do not require any settings to enable boiling/condensation.</p> <ul style="list-style-type: none"> <li>• <b>off</b>: No condensation or evaporation is modeled.</li> <li>• <b>on_gas</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.</li> <li>• <b>on_wall</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.</li> </ul> <p>Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as <b>off</b> and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.</p>





If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

#### ☉ Friction Multiplier

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the '[UserModel](#)' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

### Acceleration Options

#### Body Force Acceleration (along pipe axis)

Specify acceleration of the fluid in the flow component due to body forces such as gravity or centrifugal force.

The value entered here will impose an acceleration term on the fluid along the pipe direction, which is defined by the link (arrow) direction.





### Pressure Loss Coefficients (Bend and Taper Losses)

☒ **Determine Loss Coefficients (Fwd and Rev) from Geometry**

Specify the influences of the bends and taper losses on the Pipe pressure drop. Please note that choosing any option other than **Determine Loss Coefficients (Fwd and Rev) from Geometry** will ignore the influence of user-specified bends and tapers in pressure drop calculations.

☐ **Zero Pressure Losses from Bends and Tapers**

Calculates the pressure loss coefficient over the length of the pipe automatically based on the geometry (i.e. tapers and bends).

Neglects all pressure drop effects due to bends and tapers. Note that friction losses are computed separately.

### Flexible Wall

If the checkbox below is activated, multiple '[PipeRoundFlexWall](#)' objects will be created to represent each section of this part when discretization occurs.

☒ **External Pressure**

Pressure on the outside of the pipe walls ("def" = 1 bar). If the pipe diameter deformation is imposed with an '[XYTable](#)' this attribute is ignored.

☒ **Youngs Modulus**

Young's modulus of elasticity for the pipe wall material or reference object. Allowed reference objects include '[RLTDependenceXY](#)', '[RLTDependenceXYZ](#)', '[ProfileTransient](#)', and '[XYTable](#)'. A single Young's modulus is applied to all subvolumes along the pipe unless an '[XYTable](#)' is used. When an '[XYTable](#)' is used to specify Young's Modulus, X is defined as the interior temperature of the innermost wall layer of the pipe (as opposed to the wall's internal surface temperature) in Kelvin. The unit of the Young's modulus Y variable in the '[XYTable](#)' must be the same as the unit of the **Youngs Modulus** attribute. Note that for '[XYTable](#)' input cases if the **Wall Temperature Solver Object** in the Main folder is set to "ign" (and therefore there is an imposed constant wall temperature) there will be no change in the applied Young's modulus in time.

If the pipe diameter deformation is imposed with an '[XYTable](#)' this attribute is ignored.

☒ **Poissons Ratio**

Poisson's ratio of the pipe wall material. If the pipe diameter deformation is imposed with an '[XYTable](#)' this attribute is ignored.







## GEMShell - ShellGEMShell - Shell

This template is used to model a general volume constructed of multiple cross sections along a major direction that will be discretized into many smaller flowsplits. This is the most versatile component.

### Geometry

#### Wall Thickness

Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a ['WallThermalProperty'](#) reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the ['WallThermalProperty'](#) reference object.

#### End Type

This attribute specifies the type of ends that will be used when making the shell. The choices and their descriptions include:

- **Closed** indicates that both ends of the shell will be closed.
- **Open** indicates that both ends will be left open and will act as ports.
- **First Cross Section Open** indicates that only the first cross section will be left open to act as a port. The last cross section will be closed.
- **Last Cross Section Open** indicates that only the last cross section will be left open to act as a port. The first cross section will be closed.

#### Fitting Method

This attribute specifies the method that will be used to connect all the cross sections of the shell. The choices and their descriptions include:

- **Standard** indicates that a direct transition from one cross section to the next will be used. Essentially, this is like connecting the cross sections using a straight line to represent the shell shape.
- **Spline** indicates that a smoothing transition will be used from one cross section to the next using knowledge of upcoming cross sections. Essentially, this is like connecting the cross sections using a smooth curve to represent the shell shape.

### Cross Sections

#### Cross Section Name

Name of the cross section object(s) describing the shape to be used for the shell. The shell allows any of the 4 standard cross section shapes ([circle](#), [ellipse](#), [rectangle](#), [rounded rectangle](#)) as well as the [bi-radial](#) shape, a [custom shape](#), or an [imported shape](#). If only 1 cross section is specified, then that cross section will be used at the beginning and end of the shell. If 2 or more are specified, then the first one will be used at the beginning of the shell and the last one will be used at the end. The rest will be placed according to the **Distance to Next** below.

#### Distance to Next Cross Section

Distance to the next cross section. The cross section specified above in the same column will be extruded for this distance and blended with the next specified cross section. Only a single cross section is necessary, in







which case this value will be the total length. When multiple cross sections are specified, this is the length of each section. Also with multiple cross sections, the value of the last distance will be ignored as there is not another cross section to extrude to.

**Relative X Location of Center**

For the first cross section this attribute gives the X location of the center. For all further cross sections this attribute specifies the X location of the center relative to the center of the first cross section. The X direction is aligned with the cross section's major axis.

**Relative Y Location of Center**

For the first cross section this attribute gives the Y location of the center. For all further cross sections this attribute specifies the Y location of the center relative to the center of the first cross section. The Y direction is aligned with the cross section's minor axis.

## Location

**Location X**

Specifies the absolute X location of the component's first cross section in the [Global Coordinate System](#).

**Location Y**

Specifies the absolute Y location of the component's first cross section in the [Global Coordinate System](#).

**Location Z**

Specifies the absolute Z location of the component's first cross section in the [Global Coordinate System](#).

**Direction X**

Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

**Direction Y**

Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

**Direction Z**

Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

## Visual

**Transparency Percent**

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

**Display Color**

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue





- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

#### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

#### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025





- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

### © Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

### Additional Geometry Options

Specify optional geometric characteristics of the flow component.

### No. of Identical Flowsplits

Number of identical, parallel flowsplits represented by this individual flowsplit (usually 1.0, except for modeling many small flowsplits inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

### © Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### © Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### (©) Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).





**(☉) Wall External  
Boundary Conditions  
Object**

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

**(☉) Initial Wall  
Temperature**

Temperature of the wall at the first time step of the simulation.

**☉ Wall Temperature  
from Connected Thermal  
Primitive**

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

**☉ Adiabatic**

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

**Additional Thermal  
Options**

Specify optional thermal characteristics of the flow component.

**Heat Transfer Multiplier**

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

**Heat Input Rate**

The rate of heat input to the fluid or the name of a dependency reference object.

**Thermocouple Object**

Name of a '[Thermocouple](#)' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

**User Defined Heat  
Transfer Model**

The name of the '[UserModel](#)' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

**Condense/Evaporate  
Water Vapor (Non-  
Refrigerant Circuits)**

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to '[FluidRefrigerant](#)' Circuits, which do not require any settings to enable boiling/condensation.

- **off:** No condensation or evaporation is modeled.





- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.





### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

### User Defined Friction Model

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

## Discretization

### Discretization Scheme

Specifies which discretization values to use for this shell. The choices include:

- **Use Global Values** indicates the discretization values for this shell will be taken from the global discretization values specified in the [Export gtm](#) command.
- **Use Values Below** indicates the discretization values will be taken from the attributes below for this shell. If selected, then values specified below will override the global values specified in the [Export gtm](#) command. A value of "def" for any of the attributes below will use the global value.

### Shell Discretization Length Override Along Local X

Specifies the override target discretization length in the local X direction (X direction of the shell's [local coordinate system](#)) for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

### Shell Discretization Length Override Along Local Y

Specifies the override target discretization length in the local Y direction (Y direction of the shell's [local coordinate system](#)) for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

### Shell Discretization Length Override Along Local Z

Specifies the override target discretization length in the local Z direction (Z direction of the shell's [local coordinate system](#)) for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.



**Acceptance Ratio  
Override**

Specifies the override flowsplit accept ratio for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.





## GEMSleeve - Sleeve on a Straight or XYZ Pipe

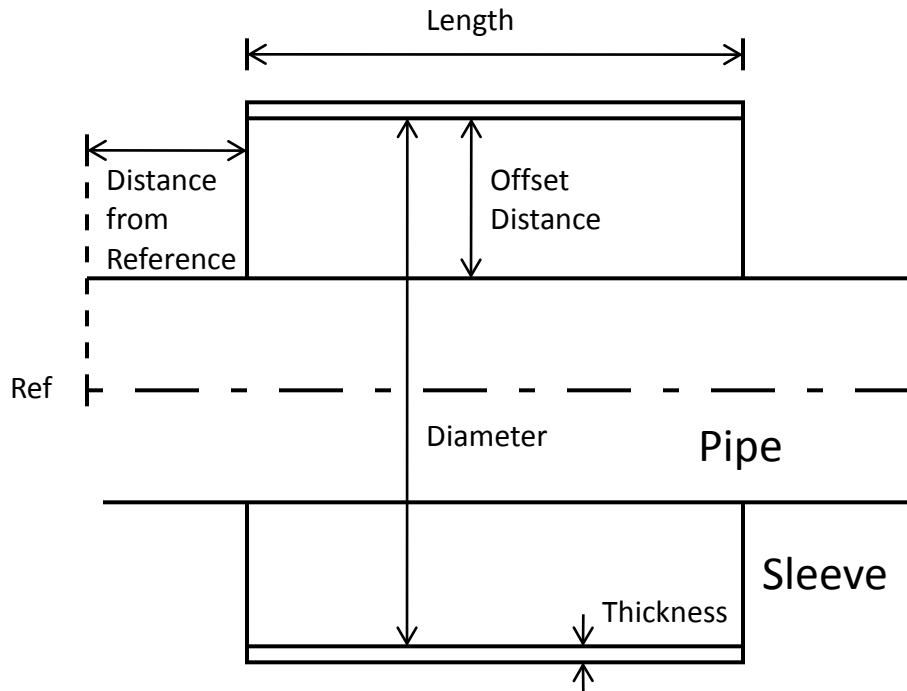
This template is used to model a sleeve that covers a straight pipe or an XYZ pipe. The volume of the sleeve will be discretized using the global shell discretization values imposed when [exporting the .gtm](#) file.

### Geometry

<b>Sleeve Type</b>	<p>Specified the method to use when creating the sleeve. Choices include:</p> <ul style="list-style-type: none"> <li>• <b>Constant Diameter</b> indicates that the <b>Diameter or Offset Distance</b> value below gives the inside diameter of the sleeve. This will result in a sleeve with a circular cross section.</li> <li>• <b>Offset</b> indicates that the <b>Diameter or Offset Distance</b> value below gives the offset distance from the outside surface of the pipe to the inside surface of the sleeve. This will result in a sleeve with a cross section that has the same shape as the parent pipe.</li> </ul>
<b>End Wall Condition</b>	<p>This attribute specifies the type of ends that will be used when making the sleeve. Choices include:</p> <ul style="list-style-type: none"> <li>• <b>Closed</b> indicates that both ends of the sleeve will be closed.</li> <li>• <b>End Open</b> indicates that only the end of the sleeve will be open. The start of the sleeve will be closed</li> <li>• <b>Start Open</b> indicates that only the start of the sleeve will be open. The end of the sleeve will be closed.</li> <li>• <b>Open</b> indicates that both ends will be left open.</li> </ul>
<b>Diameter or Offset Distance</b>	<p>Specifies the inside diameter of the sleeve or the offset distance from the outside surface of the pipe to the inside surface of the sleeve depending on the value of the <b>Sleeve Type</b> attribute above. See diagram below.</p>
<b>Sleeve Wall Thickness</b>	<p>Wall thickness of the sleeve. This will be used to draw the sleeve in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a '<a href="#">WallThermalProperty</a>' reference object. See diagram below</p>
<b>Distance from Reference</b>	<p>Specifies the distance along the pipe from the reference where the sleeve will begin. See diagram below</p>
<b>Reference</b>	<p>Specifies the reference location of the component in which to position the sleeve. Choices include:</p> <ul style="list-style-type: none"> <li>• <b>From start</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the sleeve from the start of the pipe (first cross section).</li> <li>• <b>From end</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the sleeve from the end of the pipe (last cross section).</li> </ul>
<b>Sleeve Length</b>	<p>Sleeve length. See diagram below</p>







## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then





subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

## Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

## Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

### ☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.





## Additional Geometry Options

Specify optional geometric characteristics of the flow component.

### No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### (☉) Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

### (☉) Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

### (☉) Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If





chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

### Additional Thermal Options

Specify optional thermal characteristics of the flow component.

#### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

#### Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

#### Thermocouple Object

Name of a ['Thermocouple'](#) reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

#### User Defined Heat Transfer Model

The name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

#### Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output





Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the '[UserModel](#)' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.





## GEMSleeveBend - Sleeve on a Bent Pipe

This template is used to model a sleeve that covers a bent pipe. The volume of the sleeve will be discretized using the global shell discretization values imposed when [exporting the .gtm](#) file.

### Geometry

<b>Sleeve Type</b>	<p>Specified the method to use when creating the sleeve. Choices include:</p> <ul style="list-style-type: none"> <li>• <b>Constant Diameter</b> indicates that the <b>Diameter or Offset Distance</b> value below gives the inside diameter of the sleeve. This will result in a sleeve with a circular cross section.</li> <li>• <b>Offset</b> indicates that the <b>Diameter or Offset Distance</b> value below gives the offset distance from the outside surface of the pipe to the inside surface of the sleeve. This will result in a sleeve with a cross section that has the same shape as the parent pipe.</li> </ul>
<b>End Wall Condition</b>	<p>This attribute specifies the type of ends that will be used when making the sleeve. Choices include:</p> <ul style="list-style-type: none"> <li>• <b>Closed</b> indicates that both ends of the sleeve will be closed.</li> <li>• <b>End Open</b> indicates that only the end of the sleeve will be open. The start of the sleeve will be closed</li> <li>• <b>Start Open</b> indicates that only the start of the sleeve will be open. The end of the sleeve will be closed.</li> <li>• <b>Open</b> indicates that both ends will be left open.</li> </ul>
<b>Diameter or Offset Distance</b>	<p>Specifies the inside diameter of the sleeve or the offset distance from the outside surface of the pipe to the inside surface of the sleeve depending on the value of the <b>Sleeve Type</b> attribute above.</p>
<b>Sleeve Wall Thickness</b>	<p>Wall thickness of the sleeve. This will be used to draw the sleeve in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a '<a href="#">WallThermalProperty</a>' reference object.</p>
<b>Distance from Reference</b>	<p>Specifies the distance along the pipe from the reference where the sleeve will begin.</p>
<b>Reference</b>	<p>Specifies the reference location of the component in which to position the sleeve. Choices include:</p> <ul style="list-style-type: none"> <li>• <b>From start</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the sleeve from the start of the pipe (first cross section).</li> <li>• <b>From end</b> indicates that the <b>Distance from Reference</b> value above specifies the position of the sleeve from the end of the pipe (last cross section).</li> </ul>
<b>Sleeve Length</b>	<p>Sleeve length.</p>

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0
-----------------------------	---





indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

## Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

#### ☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.





Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

☉ **Sand Roughness**

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

**Additional Geometry Options**

Specify optional geometric characteristics of the flow component.

**No. of Identical Pipes**

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

**Thermal**

**Wall Temperature Method**

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

☉ **Imposed Wall Temperature**

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

☉ **Calculated Wall**



Activates the built-in thermal wall solver to solve for the temperature of





## Temperature

the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### (☉) Wall Layer Properties Object

Name of the '[WallThermalProperty](#)' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

### (☉) Wall External Boundary Conditions Object

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

### (☉) Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

Specify optional thermal characteristics of the flow component.

### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

### Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

### Thermocouple Object

Name of a '[Thermocouple](#)' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

### User Defined Heat Transfer Model

The name of the '[UserModel](#)' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign".





## Condense/Evaporate Water Vapor (Non- Refrigerant Circuits)

The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to 'FluidRefrigerant' Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an 'EjectorConn' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity 'SensorConn' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.





## Pressure Drop

### Friction Options

#### ☉ Friction Multiplier

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.





## GEMSolidFlowVol - Solid Flow Volume

This template is used to model a general volume constructed of solid body geometry formed by an [external geometry file](#) that will be discretized into many smaller flowsplits. Unlike the shell, this volume will only be divided where custom [Child Datum Planes](#) are placed.

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the component. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>

### Main

<b>Basic Geometry and Initial Conditions</b>	Specify required geometric input as well as the initial state of the fluid in the flow component.
<b>Surface Area</b>	<p>The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:</p> $Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$ <p>where:</p> <p><math>Area</math> = <b>Surface Area</b></p> <p><math>D</math> = <b>Diameter</b> (specified above)</p> <p><math>L</math> = <b>Length</b> (specified above)</p> <p><math>D_{orifice}</math> = orifice diameters adjacent to each port</p>
<b>Initial State Name</b>	Name of the <a href="#">FluidInitialState</a> reference object describing the initial conditions inside the pipe.
<b>Surface Finish</b>	Specify surface finish characteristics of the inside of the wall in contact





with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value** : 0.0

☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

**Additional Geometry Options**

Specify optional geometric characteristics of the flow component.

**No. of Identical Pipes**

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

**Thermal**

**Wall Temperature Method**

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

☉ Imposed Wall



Impose a temperature which represents the temperature of the wall



## Temperature

surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

### ☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

### ☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

### ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

Specify optional thermal characteristics of the flow component.

### Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

### Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.





### Thermocouple Object

Name of a ['Thermocouple'](#) reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

### User Defined Heat Transfer Model

The name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

### Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an ['EjectorConn'](#) object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers,





use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the '[UserModel](#)' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.







## GEMSolidShape - General Solid Shape

This template is used to represent a shape based on solid body geometry. Solid shapes must be imported from a solid model (Native CAD formats, ACIS, STEP, or Parasolid files). This component cannot and will not be discretized into an exported model file (.gtm). To make sure this component is represented when discretized, it must first be converted into a GEM3D component using the [Convert Menu > Convert Shape to Component](#) operation. The conversion operation can be undone using the [Convert Menu > Deconvert Component](#) operation as long as the "Allow Mesh Deconversion" option is turned on (checked) in [File>Options>Conversion](#). Solid shapes can also be converted into [GEMMeshShape - General Mesh Shape](#) using the [Convert Menu > Convert Solid to Mesh](#) operation.

---

## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the component. The color choices include: <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>





## GEMSolidShell - Solid Shell

This template is used to model a general volume constructed of solid body geometry formed by an [Import 3D - Import 3D file](#) that will be discretized into many smaller flowsplits.

### Orientation

*The attributes on this tab specify the location and orientation of the [local coordinate system \(LCS\)](#) that will be used as the reference during feature creation (baffles, etc.) and discretization. When it is necessary to change the location and orientation of the component's LCS, it is recommended to use the apply button in the component often to see a visual update. When doing this the [Component's Axis](#) should be shown, which can be done from the right-click menu for the component.*

<b>X Location of LCS</b>	Specifies the absolute X location of the component's LCS in the <a href="#">Global Coordinate System</a> .
<b>Y Location of LCS</b>	Specifies the absolute Y location of the component's LCS in the <a href="#">Global Coordinate System</a> .
<b>Z Location of LCS</b>	Specifies the absolute Z location of the component's LCS in the <a href="#">Global Coordinate System</a> .
<b>X Direction of LCS</b>	Specifies the X component of the vector describing the direction that the component's major axis will be pointed. Only a unit vector is needed to describe the direction, so the <b>X Direction of LCS</b> , <b>Y Direction of LCS</b> , and <b>Z Direction of LCS</b> attributes may be replaced with the equivalent unit vector.
<b>Y Direction of LCS</b>	Specifies the Y component of the vector describing the direction that the component's major axis will be pointed. Only a unit vector is needed to describe the direction, so the <b>X Direction of LCS</b> , <b>Y Direction of LCS</b> , and <b>Z Direction of LCS</b> attributes may be replaced with the equivalent unit vector.
<b>Z Direction of LCS</b>	Specifies the Z component of the vector describing the direction that the component's major axis will be pointed. Only a unit vector is needed to describe the direction, so the <b>X Direction of LCS</b> , <b>Y Direction of LCS</b> , and <b>Z Direction of LCS</b> attributes may be replaced with the equivalent unit vector.
<b>Reference Direction Angle</b>	Specifies the rotational orientation of the LCS. The angle specified here is used to rotate the orientation of the LCS from the default calculated values given by the following 3 reference direction attributes. A good practice is to show the component's axis in the graphical window (right-click on component→Component's Axis) and modify this angle until the desired orientation is reached (use the Apply button in the component to visually see the changes).
<b>Reference Direction X</b>	Specifies the X component of the vector describing the reference direction. The reference direction gives the direction of the X axis of the LCS. It is highly recommended to use a value of "def", letting the application determine this value. When using "def", it must be used for





all reference direction attributes (X, Y, and Z) together (or none).

*Only a unit vector is needed to describe the reference direction, so the **Reference Direction X**, **Reference Direction Y**, and **Reference Direction Z** attributes may be replaced with the equivalent unit vector.*

#### Reference Direction Y

Specifies the Y component of the vector describing the reference direction. The reference direction gives the direction of the X axis of the LCS. It is highly recommended to use a value of "def", letting the application determine this value. When using "def", it must be used for all reference direction attributes (X, Y, and Z) together (or none).

*Only a unit vector is needed to describe the reference direction, so the **Reference Direction X**, **Reference Direction Y**, and **Reference Direction Z** attributes may be replaced with the equivalent unit vector.*

#### Reference Direction Z

Specifies the Z component of the vector describing the reference direction. The reference direction gives the direction of the X axis of the LCS. It is highly recommended to use a value of "def", letting the application determine this value. When using "def", it must be used for all reference direction attributes (X, Y, and Z) together (or none).

*Only a unit vector is needed to describe the reference direction, so the **Reference Direction X**, **Reference Direction Y**, and **Reference Direction Z** attributes may be replaced with the equivalent unit vector.*

## Discretization

#### Discretization Scheme

Specifies which discretization values to use for this shell. The choices include:

- **Use Global Values** indicates the discretization values for this shell will be taken from the global discretization values specified in the [Export gtm](#) command.
- **Use Values Below** indicates the discretization values will be taken from the attributes below for this shell. If selected, then values specified below will override the global values specified in the [Export gtm](#) command. A value of "def" for any of the attributes below will use the global value.

#### Shell Discretization Length Override Along Local X

Specifies the override target discretization length in the local X direction (X direction of the shell's [local coordinate system](#)) for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

#### Shell Discretization Length Override Along Local Y

Specifies the override target discretization length in the local Y direction (Y direction of the shell's [local coordinate system](#)) for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.





### Shell Discretization Length Override Along Local Z

Specifies the override target discretization length in the local Z direction (Z direction of the shell's [local coordinate system](#)) for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

### Acceptance Ratio Override

Specifies the override flowsplit acceptance ratio for this shell. "def" may be entered for this attribute. If **Use Global Values** is selected above, then "def" means this attribute will not be used. If **Use Values Below** is selected above, then "def" means use the global attribute specified in the [Export gtm](#) command.

## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the component. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)





	<p><math>L</math> = <b>Length</b> (specified above)</p> <p><math>D_{orifice}</math> = orifice diameters adjacent to each port</p>
<b>Initial State Name</b>	Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.
<b>Surface Finish</b>	Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.
<p><input checked="" type="radio"/> <b>Smooth</b></p> <p><input type="radio"/> <b>Roughness from Material</b></p>	<p>Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).</p> <p>This attribute gives choices of materials that may be used to specify the surface roughness.</p> <p><u>Material Name : Default Roughness Value (mm)</u></p> <ul style="list-style-type: none"> <li>• <b>drawn_metal</b> : 0.002</li> <li>• <b>steel</b> : 0.046</li> <li>• <b>cast_iron</b> : 0.26</li> <li>• <b>light_rust_steel</b> : 0.25</li> <li>• <b>heavy_rust_steel</b> : 1.0</li> <li>• <b>smooth_plastic</b> : 0.0025</li> <li>• <b>smooth_rubber</b> : 0.025</li> <li>• <b>smooth_galvanized</b> : 0.025</li> <li>• <b>normal_galvanized</b> : 0.15</li> <li>• <b>wrought_iron</b> : 0.046</li> <li>• <b>asphalted_cast</b> : 0.12</li> <li>• <b>extruded_aluminum</b> : 0.003</li> <li>• <b>user_value</b> : 0.0</li> </ul>
<input checked="" type="radio"/> <b>Sand Roughness</b>	<p>Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.</p> <p>A numeric value or a parameter may be entered for this attribute.</p>
<b>Additional Geometry Options</b>	Specify optional geometric characteristics of the flow component.
<b>No. of Identical Pipes</b>	Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)





## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

#### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

#### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

#### ☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

#### ☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

#### ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

#### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

#### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

### Additional Thermal Options

Specify optional thermal characteristics of the flow component.





<b>Heat Transfer Multiplier</b>	Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for <b>Surface Finish</b> , additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")
<b>Heat Input Rate</b>	The rate of heat input to the fluid or the name of a dependency reference object.
<b>Thermocouple Object</b>	Name of a <a href="#">'Thermocouple'</a> reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe* or FlowSplit* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.
<b>User Defined Heat Transfer Model</b>	The name of the <a href="#">'UserModel'</a> object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. If a user model is not going to be used this attribute should be set to "ign". The heat transfer coefficient value that is calculated through the user model will also be multiplied by the <b>Heat Transfer Multiplier</b> value.
<b>Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)</b>	<p>Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to <a href="#">'FluidRefrigerant'</a> Circuits, which do not require any settings to enable boiling/condensation.</p> <ul style="list-style-type: none"> <li>• <b>off</b>: No condensation or evaporation is modeled.</li> <li>• <b>on_gas</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.</li> <li>• <b>on_wall</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.</li> </ul> <p>Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as <b>off</b> and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.</p> <p>If this flag is set to <b>on_gas</b> or <b>on_wall</b> the properties of both vapor and liquid water must be defined.</p>







Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

#### ☉ Friction Multiplier

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the '[UserModel](#)' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.





## GEMSPipe - Straight Pipe

This template is used to model straight pipes. This pipe can have any shape cross section and allows the cross section to change over the length of the pipe.

### Geometry

<b>Wall Thickness</b>	Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a 'WallThermalProperty' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the 'WallThermalProperty' reference object.
-----------------------	--

### Cross Sections

<b>Cross Section Name</b>	Name of the cross section object(s) describing the shape to be used for the straight pipe. The straight pipe allows any of the 4 standard cross section shapes ( <a href="#">circle</a> , <a href="#">ellipse</a> , <a href="#">rectangle</a> , <a href="#">rounded rectangle</a> ) as well as a <a href="#">custom shape</a> . If only 1 cross section is specified, then that cross section will be used at the beginning and end of the pipe. If 2 or more are specified, then the first one will be used at the beginning of the pipe and the last one will be used at the end. The rest will be placed according to the Distance to Next below. The resulting pipe will use a smooth transition to change cross sections along the length of the pipe.
<b>Diameter</b>	If the diameter option is selected, then the pipe will always have a round cross section using the specified diameters. At least 2 diameters must be defined for the pipe, if more than 2 diameters are specified they will be placed according to the Distance to Next below. The resulting pipe will taper between cross sections of varying diameter.
<b>Distance to Next Cross Section</b>	Distance to the next cross section. The cross section specified above in the same column will be extruded for this distance and blended with the next specified cross section. Only a single cross section is necessary, in which case this value will be the pipe length. When multiple cross sections are specified, this is the length of each section. Also with multiple cross sections, the value of the last distance will be ignored as there is not another cross section to extrude to.

### Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the





component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

#### Direction Y

Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

#### Direction Z

Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the **Direction X**, **Direction Y**, and **Direction Z** attributes may be replaced with the equivalent unit vector.

### Visual

#### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

#### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

### Main

#### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Discretization Length

Discretization length to be used for pipes during the discretization process. This length does not need to be an even fraction of the entire pipe length; the code will adjust to divide the pipe appropriately. A value of "def" will use the discretization length found in the global discretization window. See [export gtm](#) for additional information.

#### Initial State Name

Name of the '[FluidInitialState](#)' reference object describing the initial conditions inside the pipe.

#### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.





☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).

☉ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

☉ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

**Additional Geometry Options**

Specify optional geometric characteristics of the flow component.

**Pipe Elevation Change or 3D Acceleration Object**

This attribute defines either the elevation change of the pipe from the inlet to the outlet, or 3D acceleration resolved along the pipe axis via a '[PipeMotion3DBM](#)' reference object. If "def" is entered, then the elevation change or 3D acceleration object of the pipe will be automatically generated from the option selected in the **Gravity/Acceleration Options** field of [File>Options>Discretization](#).

When the gravity option is used, a positive value means port 2 is at a higher elevation than port 1, and vice versa. The acceleration due to gravity is assumed to be 9.80665 m/s<sup>2</sup>. When '[XYTable](#)' reference object is used, X is defined as pipe length (which is normalized) and Y is the elevation change.

This attribute cannot be used in parallel with the **Body Force Acceleration** attribute under the Options folder.

**No. of Identical Pipes**

Number of identical, parallel pipes represented by this pipe (usually 1.0,





except for modeling many small pipes inside some heat-exchangers).  
("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

#### ⊙ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. '[XYTable](#)' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

#### ⊙ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

#### (⊙) Wall Layer Properties Object

Name of the '[WallThermalProperty](#)' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

#### (⊙) Wall External Boundary Conditions Object

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

#### (⊙) Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

#### ⊙ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses). This option should be used when this Pipe object will be an intake or exhaust port for a cylinder.

#### ⊙ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not





## Additional Thermal Options

### Heat Transfer Multiplier

to the wall heat transfer rate).

Specify optional thermal characteristics of the flow component.

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

### Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

### Thermocouple Object

Name of a ['Thermocouple'](#) reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.

### ☉ Heat Transfer Correlation (Colburn)

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

### ☉ User Defined Heat Transfer Model

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

### ☉ Heat Transfer Coefficient

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

### Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated





(typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction



The name of the '[UserModel](#)' object which will be used to calculate the



## Model

Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.

## Acceleration Options

Specify acceleration of the fluid in the flow component due to body forces such as gravity or centrifugal force.

## Body Force Acceleration (along pipe axis)

The value entered here will impose an acceleration term on the fluid along the pipe direction, which is defined by the link (arrow) direction.

## Pressure Loss Coefficients (Bend and Taper Losses)

Specify the influences of the bends and taper losses on the Pipe pressure drop. Please note that choosing any option other than **Determine Loss Coefficients (Fwd and Rev) from Geometry** will ignore the influence of user-specified bends and tapers in pressure drop calculations.

### ☐ Determine Loss Coefficients (Fwd and Rev) from Geometry

Calculates the pressure loss coefficient over the length of the pipe automatically based on the geometry (i.e. tapers and bends).

### ☐ Zero Pressure Losses from Bends and Tapers

Neglects all pressure drop effects due to bends and tapers. This is equivalent to specifying **Forward Loss Coefficient** and **Backward Loss Coefficient** as zero. Note that friction losses are computed separately.

### ☐ Forward Pressure Loss Coefficient

Pressure loss coefficient for flow from port 1 to port 2. This attribute is used to account for pressure losses due to geometry, such as bends and non-circular cross-sections. Enter "def" to have the pressure loss automatically calculated by the code using the cross-sectional shape and bend information that is entered. (0 indicates no additional pressure loss.)

### ☒ Reverse Pressure Loss Coefficient

Pressure loss coefficient in the opposite direction from port 2 to port 1 (see **Forward Pressure Loss Coefficient** above).

## Flexible Wall

If the checkbox below is activated, a **PipeRoundFlexWall** object will be created to represent this part when discretization occurs.

### ☒ External Pressure

Pressure on the outside of the pipe walls ("def" = 1 bar). If the pipe diameter deformation is imposed with an **XYTable** this attribute is ignored.

### ☒ Youngs Modulus

Young's modulus of elasticity for the pipe wall material or reference object. Allowed reference objects include **RLTDependenceXY**, **RLTDependenceXYZ**, **ProfileTransient**, and **XYTable**. A single Young's modulus is applied to all subvolumes along the pipe unless an **XYTable** is used. When an **XYTable** is used to specify Young's Modulus, X is defined as the interior temperature of the innermost wall layer of the pipe (as opposed to the wall's internal surface temperature) in Kelvin. The unit of the Young's modulus Y variable in the **XYTable** must be the same as the unit of the **Youngs Modulus** attribute. Note that for **XYTable** input cases if the **Wall Temperature Solver Object** in the Main folder is set to "ign" (and therefore there is an







imposed constant wall temperature) there will be no change in the applied Young's modulus in time.

If the pipe diameter deformation is imposed with an 'XYTable' this attribute is ignored.

**(☒) Poissons Ratio**

Poisson's ratio of the pipe wall material. If the pipe diameter deformation is imposed with an 'XYTable' this attribute is ignored.







## GEMSubAssExternalConn - External Subassembly Connection

This template is used to specify the details for the external subassembly connection. The port number for connections and the flow direction can be specified here.

### Main

#### Port ID

Specifies the port ID number to be used for the external subassembly connection during the discretization routine. This value will get copied directly into the **Corresponding Subassembly Port #** attribute in the '[SubAssExternalConn](#)' object that is created in the GT-SUITE model file. This value can be any positive integer.

#### Flow Direction

Specifies the desired connection direction to be used at the external subassembly connection. This direction will be used by the discretization routine to determine the connection direction in the rest of the model. The choices include:

- **def** Specifies that GEM3D will determine the connection direction during discretization.
- **Inlet** Specifies that the connection will point from the subassembly connection into the flow component. This should be chosen for an inlet boundary condition.
- **Outlet** Specifies that the connection will point from the flow component into the subassembly connection. This should be chosen for an outlet boundary condition.

### Visual

#### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

#### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black





## GEMTank3D - Tank with 3-Dimensional Acceleration

This object is used to model a fluid reservoir or tank being subjected to a user defined 3-dimensional acceleration. An .stl file will be created, representative of the tank geometry, when discretizing 'GEMTank3D' to GT-ISE.

### Main

<b>Initial Conditions</b>	Specify the initial conditions of the tank's fluids.
<b>Liquid Initial State Name</b>	Name of the 'FluidInitialState' reference object describing the initial conditions of the liquid in the tank.
<b>Gas/Vapor Initial State Name</b>	Name of the 'FluidInitialState' reference object describing the initial conditions of the gas/vapor in the tank.
<b>☉ Initial Liquid Volume</b>	Define the initial volume occupied by the liquid as a volume. The gas/vapor occupies the remaining volume.
<b>☉ Initial Liquid Volume Fraction</b>	Define the initial volume occupied by the liquid as a volume fraction. The gas/vapor occupies the remaining volume.

### Thermal Behavior

<b>☉ Imposed Wall Temperature</b>	Specify a temperature or the name of a reference object which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this <b>Imposed Wall Temperature</b> , along with the fluid temperature.
<b>☉ Model Wall with Connected External Thermal Primitives</b>	If this option is selected, the heat transfer rate from the fluid to the wall will be modeled through the use of thermal primitives that connect to the tank part. The thermal primitives will need to be created by the user.

### Fill Level Gauges

This folder allows you to measure the liquid level between two points - a full point and an empty point. A line will be created between these two points and the fill level will be equal to the fraction of this line that is covered by the liquid. If no Fill Level Gauges are required please set each attribute in this folder to 'ign'.

<b>Fill Level Gauge ID Number</b>	User defined identifier for each of the tank's gauges. Each gauge number must be a unique positive integer.
<b>Define Point at Which Gauge Reads Empty</b>	Specify the location of the gauge empty point. If the liquid level is below this point, the gauge will indicate the tank is empty.
<b>X-Coordinate of Gauge Empty Point</b>	The x-coordinate of the empty point.
<b>Y-Coordinate of Gauge Empty Point</b>	The y-coordinate of the empty point.
<b>Z-Coordinate of Gauge Empty Point</b>	The z-coordinate of the empty point.





### Define Point at Which Gauge Reads Full

Specify the location of the gauge full point. If the liquid level is at or above this point, the gauge will indicate that the tank is full.

#### X-Coordinate of Gauge Full Point

The x-coordinate of the full point.

#### Y-Coordinate of Gauge Full Point

The y-coordinate of the full point.

#### Z-Coordinate of Gauge Full Point

The z-coordinate of the full point.

## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black





## GEMTank3DPort - Port for Tank3D

This template is used to add a port to 'GEMTank3D'. The port can be located inside of or on the surface of the tank.

### Main

<b>Port Label</b>	User defined identifier for the tank's ports. Each port must have its own unique label.
<b>Surface Area</b>	The surface area of the tank's port.

### Visual

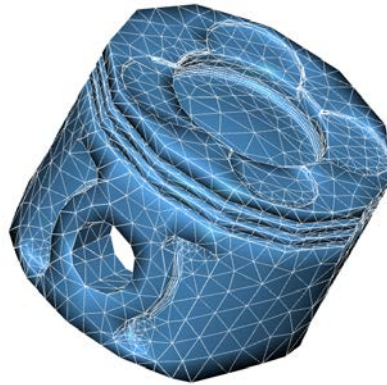
<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>





## GEMThermalFE – 3D Finite Element Thermal Mass

This template is used to represent thermal mass using a 3D finite element component meshed from 3D solid geometry. GEM3D includes a built-in meshing tool which can generate this component when accessed via the standard “Convert Shape to Component” function. Alternatively, this component is also created when converting an external imported 3D mesh ([GEMFEVolumeMesh](#)) to a GEM FE component. This 3DFE approach can be used in any case where a lumped mass approximation is not deemed appropriate and more resolution is desired.



Upon export, the GT model file will include a ‘[ThermalFiniteElement](#)’ part which will call an ‘[FEMesh3D](#)’ reference object that contains all of the node, element, and surface port data. This part may be connected directly to fluid volumes or other boundary conditions. If the end goal is to use the 3D mesh within the FE Cylinder Structure templates (either ‘[FECylinderStructure](#)’ or ‘[FECylStrucCustom](#)’), then it will be necessary to copy the resulting ‘[FEMesh3D](#)’ reference object from the exported model into the desired model in GT-ISE.

After converting the solid shape to the FE mesh, surface ports may be defined by right clicking on the component and selecting **Thermal FE Port ...**. See the help for ‘[GEMThermalFEPort](#)’ for more details.

### Main

#### FE Mesh Reference Object

The name of a [FEMesh3D](#) reference object which contains all of the node and element data for the mesh. This object is generated automatically by GEM3D during the meshing operation or the conversion of an external imported 3D mesh. Note that some of the data in this ref. object is completed by GEM3D upon export of the GT model file based on the presence of children thermal FE ports.

#### Initial Temperature

Temperature of the thermal mass at the start of simulation.

#### Material Properties Object (for all FE Elements)

The name of a [MaterialThermalProp](#) reference object which contains the material properties of the FE mesh. These properties will be assigned to all elements in the mesh.

Note: After exporting the model, it is possible to manually edit the resulting objects to define multiple materials if using the ‘[ThermalFiniteElement](#)’ template.





## Visual

---

### Transparency Percent

Indicates the transparency level used when drawing the component. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

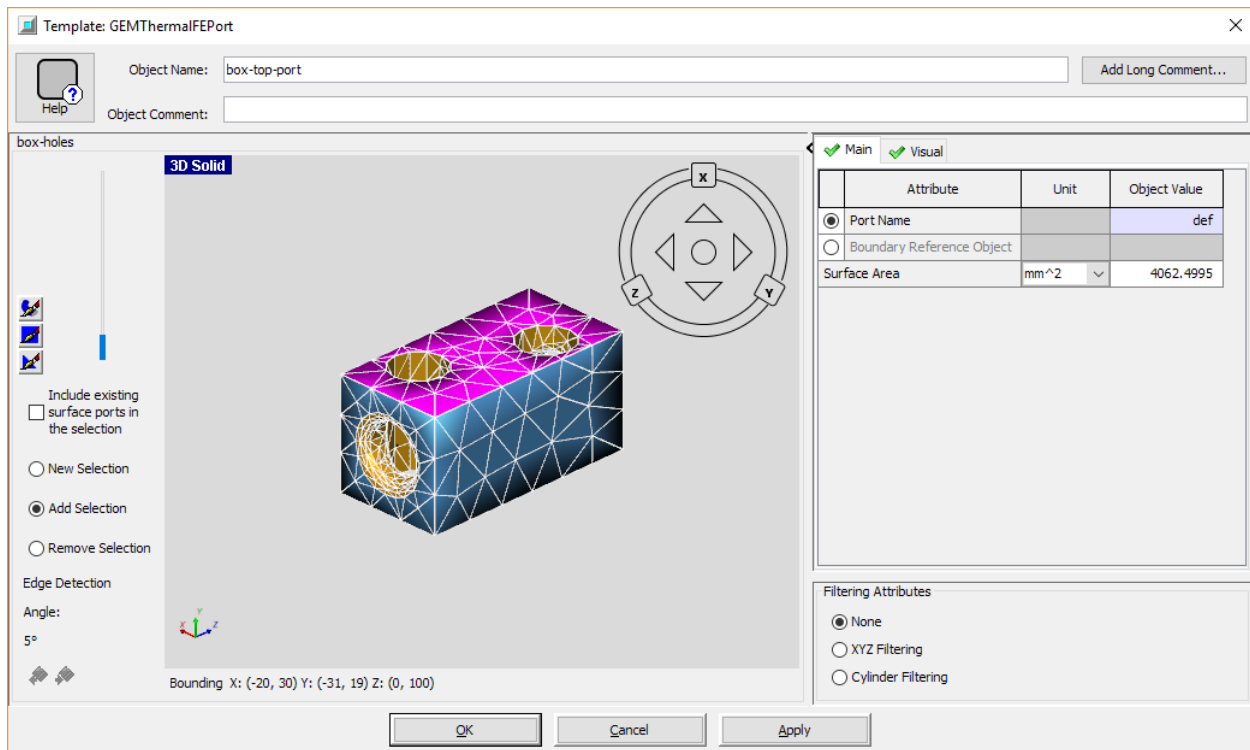
Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black



## GEMThermalFEPort – Surface Port on 3D FE Mesh

This feature exists as a child reference object belonging to a 'GEMThermalFE' component, and serves to define a surface area on the 3D mesh which will eventually connect to another volume, mass, or boundary condition for heat transfer (conduction, convection, radiation). A Thermal Port may be added to a 3D mesh by right clicking on the mesh and selecting **Thermal FE Port ...**. The resulting window allows the surface of the mesh to be “painted”.



The options on the left-hand side control whether a mouse-click on the mesh will add or remove triangles from the selection. The option “Include existing surface ports in the selection” controls whether a given triangle can be assigned to multiple surface ports. If this box is unselected, any triangle that has been previously painted and assigned to an already existing surface port will not be possible to select.

The slider on the left controls the “Edge Detection Angle”. The minimum value of 0 degrees will result in a mouse-click painting only the triangle that was clicked on. As the angle increases, the painting will extend to adjacent triangles until the angle of the edge between triangles exceeds the threshold.

On the right hand side, there are options for filtering the triangles to be painted by either a box (XYZ filtering) or a cylinder. This can be used to selectively paint portions of a surface which does not have an edge angle at the boundary of the surface to make selection easy.

### Main

#### Port Name

If the surface port will eventually connect to a fluid volume or thermal mass part, then a port name should be defined directly. A 'def' value will use the name of the GEMThermalFEPort for the surface name in the



FEMesh3D and ThermalFiniteElement in GT-ISE

**Boundary Reference Object**

If a convection thermal boundary condition should be applied, then a 'ThermalBound' reference object may be defined to specify the bounding temperature and heat transfer coefficient.

**Surface Area**

The surface area of the selection. This cell is filled in automatically as the surface is painted in the graphical window. The cell is editable, so the value can be manually overridden.

---

**Visual**

---

**Transparency Percent**

Indicates the transparency level used when drawing the component. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

**Display Color**

Indicates the color used when drawing the part. The color choices include:

- **Red**
- **Blue**
- **Dark Blue**
- **Green**
- **Copper**
- **Gold**
- **Grey**
- **Black**







## GEMThermalMass - Thermal Mass

This template is used to describe a thermal mass object from 3D solid geometry. The volume is calculated from the converted shape, and the material properties are used along with the volume to calculate the mass of the part. Child datum planes ([GEMDatumPlane](#)) can be used to divide the GEMThermalMass; this will create connected thermal masses when the model is exported. See the **Notes** section at the bottom for more details.

### Main

<b>Material Properties Object</b>	The name of a ' <a href="#">MaterialThermalProp</a> ' reference object which will be used to calculate the specific heat and thermal conductivity of the object.
<b>Volume</b>	The volume of the shape. This is measured during the conversion to a GEMThermalMass.
<b>Reference Temperature for Converting Volume to Mass</b>	This temperature will be used when exporting the model to determine the material density. The mass of the ThermalMass will be calculated from this material density and the calculated volume.
<b>Initial Temperature</b>	Temperature of the thermal mass at the start of simulation.
<b>Source Heat Rate</b>	Heat Rate to be imposed on the mass as an energy "source term" ("ign" = 0).
<b>Emissivity</b>	The emissivity of the material is specified here. Ignore ("ign") is available if a ' <a href="#">MaterialThermalProp</a> ' is not connected to the port. An ' <a href="#">MaterialThermalProp</a> ' can be used to specify the emissivity as a function of the port temperature (in K).

### Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li><li>• Dark Blue</li><li>• Green</li><li>• Copper</li><li>• Gold</li><li>• Grey</li><li>• Black</li></ul>

**Notes:** The GEMThermalMass must be created from a complete solid shape. This is necessary for an accurate volume and surface area calculation of each discretized thermal mass. If there are problems with





the imported geometry or problems obtaining the correct file format, GT-SPACECLAIM can be used. The file could be translated from various file formats into a supported format and any solid or surface flaws could be fixed.

Child Datum Planes can be added (from the right-click menu or Builder → Add Datum Plane → Child Datum Plane...) to discretize the GEMThermalMass into smaller segments. These datum planes will extend all the way through the component, regardless of their displayed size. The datum planes must be positioned in 3D space using absolute coordinates, not relative to the thermal mass location.

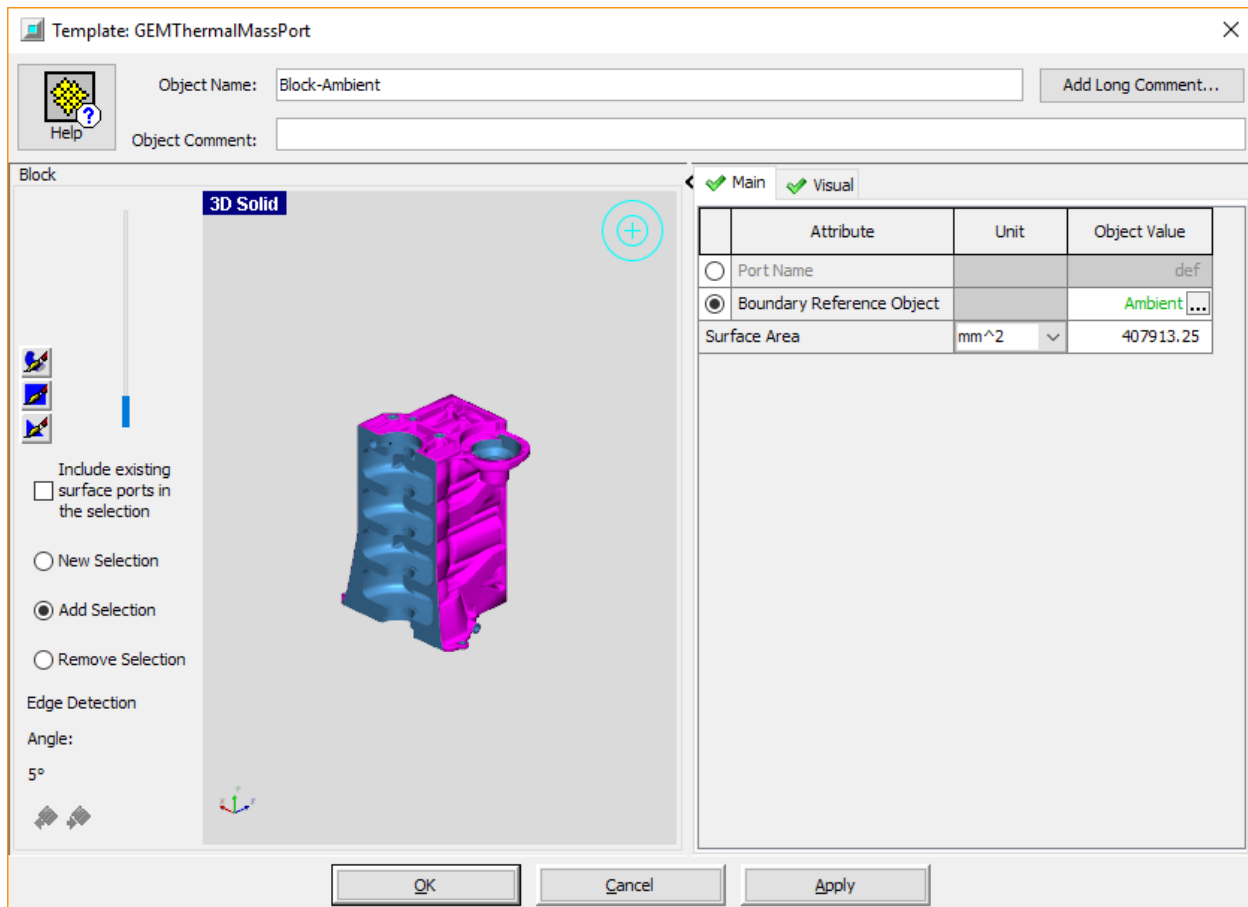
When the GEMThermalMass model is exported, several additional calculations will take place. The first is a mass calculation. The mass of the created ThermalMass in the gtm file is determined from the measured **Volume** of the GEMThermalMass, and the density of the material calculated at the Reference Temperature for Converting Volume to Mass. If child datum planes are added to the GEMThermalMass, the entire shape will be sliced using the datum planes and the volume of each section will be calculated. Individual thermal mass components will be created for each section of the GEMThermalMass, with the mass of these components calculated by their respective volumes. In addition to the mass calculation, the Ports folder of the Thermal Mass will also be filled out. The Distance to Mass Center will be calculated from each section's center of mass to the child datum plane. The Cross Section Area will be calculated from measuring the cut area for each datum plane. The resulting model is a network of thermal masses, connected by ConductanceConns.

Connections to external boundary conditions and other thermal masses can be made using the Thermal Mass Port, conduction connection, and connection connections respectively.



## GEMThermalMassPort – Surface Port of a Thermal Mass

This feature exists as a child reference object belonging to a GEMThermalMass component, and serves to define a surface area on the mass which will eventually connect to another volume, mass, or boundary condition for heat transfer (conduction, convection, radiation). A Thermal Port may be added to a thermal mass by right clicking on the mass and selecting **Thermal Mass Port ...**. The resulting window allows the surface of the mass to be “painted”.



The options on the left-hand side control whether a mouse-click on the mesh will add or remove triangles from the selection. The option “Include existing surface ports in the selection” controls whether a given triangle can be assigned to multiple surface ports. If this box is unselected, any triangle that has been previously painted and assigned to an already existing surface port will not be possible to select.

The slider on the left controls the “Edge Detection Angle”. The minimum value of 0 degrees will result in a mouse-click painting only the triangle that was clicked on. As the angle increases, the painting will extend to adjacent triangles until the angle of the edge between triangles exceeds the threshold.

### Main

#### Port Name

If the surface port will later (after the GT model is exported) be connected manually to a fluid volume or thermal mass part, then a port



## GEMThermalMassPort Reference Object

name may be defined directly. If 'def' is entered, the port name will be taken from the GEMThermalMassPort name. Note that the name specified here will be used as the **Port Name** in the resulting ThermalMass and may not be the actual port number (**Boundary #**) as shown in the screenshot below:

<div> <span>✓ Main</span> <span>✓ Ports</span> <span>✗ Plots</span> </div>				
Attribute	Unit	Boundary #1	Boundary #2	Boundary #3
Port Name		1...	2...	100...
Distance to Mass Center	mm	108.0549...	115.2771...	116.6186...
Cross Sectional Area	mm^2	659.5154...	58246.54...	130747.0...
Emissivity	fraction	0.6...	0.6...	0.6...

A value of “def” may be entered. In this case, GEM will automatically define the port using an available integer number.

### Boundary Reference Object

To apply a convection boundary condition, a 'ThermalBound' reference object may be defined to specify the bounding temperature and heat transfer coefficient.

### Surface Area

The surface area of the selection. This cell is filled in automatically as the surface is painted in the graphical window. The cell is editable, so the value can be manually overridden.

## Visual

### Transparency Percent

Indicates the transparency level used when drawing the component. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent. A value of "def" will use the value specified with the **Default Transparency Percentage** option in File → Options → General.

### Display Color

Indicates the color used when drawing the part. The color choices include:

- Red
- Blue
- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black





## GEMTsplit - T-Shaped Flowsplit

This template is used to describe a cylindrical shaped flowsplit volume connected to three flow components in a T-configuration.

### Geometry

<b>Diameter</b>	Diameter of the cylindrical volume. This will also be the diameter used to draw the part in the graphical window.
<b>Length</b>	Length of the cylindrical volume.
<b>Diameter of Port 3</b>	Diameter of the perpendicular port (port 3). A value of "def" will set this diameter equal to the <b>Diameter</b> attribute above. This value will <u>only</u> be used to draw the perpendicular port in the graphical window. The actual diameter of the perpendicular port will be determined from the connected pipe during discretization.
<b>Wall Thickness</b>	Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a ' <a href="#">WallThermalProperty</a> ' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the ' <a href="#">WallThermalProperty</a> ' reference object.

### Location

<b>Location X</b>	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Y</b>	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Location Z</b>	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z</b>	Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.





## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

#### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

#### Initial State Name

Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

#### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).





### ◎ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

### ◎ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

### Additional Geometry Options

Specify optional geometric characteristics of the flow component.

### No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

### ◎ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall





## ☉ Calculated Wall Temperature

temperature distribution, where X is normalized length with range 0 to 1.

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

## ☉ Wall Layer Properties Object

Name of the '[WallThermalProperty](#)' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

## ☉ Wall External Boundary Conditions Object

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

## ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

## ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

## ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

## Additional Thermal Options

Specify optional thermal characteristics of the flow component.

## Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

## Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

## Thermocouple Object

Name of a '[Thermocouple](#)' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.





**☉ Heat Transfer  
Correlation (Colburn)**

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

**☉ User Defined Heat  
Transfer Model**

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

**☉ Heat Transfer  
Coefficient**

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

**Condense/Evaporate  
Water Vapor (Non-  
Refrigerant Circuits)**

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an ['EjectorConn'](#) object if





desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

### Flowsplit HTC Calculation Method

#### Method #1 (v7.5 and prior)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, and then they are weighted by the boundary area. This method tends to under predict the heat transfer coefficient in the flowsplit, and is only recommended to reproduce previous version results.

#### Method #2 (Recommended)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port. The resulting coefficients are averaged based on the boundary flow rates. This method produces a more accurate prediction for the heat transfer coefficient, and is recommended especially in applications where the heat transfer in the flowsplit is a significant part of the overall circuit heat transfer rate.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☉ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☉ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the '[UserModel](#)' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction**





**Multiplier** value.






## GEMVanePump – Detailed Vane Pump Model Template

This template is used to generate a detailed vane pump flow model from an existing CAD file of a vane pump. In order to use this template, a CAD file must be supplied for the vane pump and must contain the shapes representing the outer stator, inner rotor, vane, inlet port volume(s), and outlet port volume(s). By identifying these shapes, GEM will be able to generate a 1D model of the pump, with most attribute values automatically calculated, and all parts automatically connected.

There are a few points to consider when using the vane pump generation feature, including:

- 1) The vane pump CAD model should be supplied at maximum eccentricity if possible, as the mechanical assembly assumes this condition. If a mechanical assembly is used and the CAD file is NOT at max. eccentricity, the max. eccentricity and eccentricity angle at max. eccentricity attributes in the mechanical compound must be manually modified by the user. Furthermore, the CAD model should *NOT* be supplied at zero eccentricity, as GEM has no way of extrapolating to the proper eccentricity angle away from the zero eccentricity position.
- 2) The rotor is assumed to be perfectly circular, and if it is not circular, GEM will assume its diameter is based on the maximum value. Furthermore, the dead volume of fluid due to the non-circular rotor shape must be manually added by the user in the 'VaneShaft' template in GT-ISE after exporting to the .gtm file.
- 3) The port area of a groove is not currently calculated. Any groove area profile must be manually entered by the user in the 'VaneShaft' template in GT-ISE after exporting to the .gtm file.
- 4) If using a sliding style pump with two control chambers, only 1 control chamber is currently allowed, and the second chamber must manually be defined in GT-ISE after exporting to the .gtm file.

Before selecting the geometry shapes in this template, it is recommended to first name the shapes by simply double-clicking on each shape in GEM and changing the name to something appropriate. This will make it easy to identify the proper shape in the GEMVanePump template simply by selecting the  button for each shape attribute and navigating to the shape name.

### Geometry Shapes

<b>Outer Stator Shape</b>	Name of the mesh shape that represents the outer stator. See the schematic below showing the stator shape.
<b>Inner Rotor Shape</b>	Name of the mesh shape that represents the inner rotor. See the schematic below showing the rotor shape.
<b>Vane Shape</b>	Name of the mesh shape that represents the vane. See the schematic below showing the vane.
<b>Inlet Port 1 Shape</b>	Name of the mesh shape that represents the first inlet port. See the schematic below showing the first inlet port.
<b>Outlet Port 1 Shape</b>	Name of the mesh shape that represents the first outlet port. See the schematic below showing the first outlet port.
<b>Inlet Port 2 Shape</b>	Name of the mesh shape that represents the second inlet port.
<b>Outlet Port 2 Shape</b>	Name of the mesh shape that represents the second outlet port.





## Mechanical Setup

### Model Mechanical Assembly for Dynamic Eccentricity?

Option to consider the mechanical assembly for dynamic prediction of eccentricity. Setting the option to "NO" will assume a fixed eccentricity, while setting the option to "YES" will require user input for calculation of the dynamic eccentricity.

☐ NO

Selecting this option will assume a fixed eccentricity for the pump.

☒ YES

Selecting this option will create a mechanical assembly and will require user input in the attributes below for calculation of the dynamic eccentricity.

### (☒) Vane Pump Type

The user has two options:

- **Sliding** – this will model a sliding style pump, where the stator is assumed to only translate with respect to the rotor
- **Pivoting** – this will model a pivoting style pump, where the stator is assumed to pivot about a pivot point.

### Pivot Center Control Point (if Pivoting Type)

The name of the control point representing the pivot center point. This attribute applies only to pivoting style pumps, and should be set to "ign" for sliding style pumps. See the schematic below illustrating this location.

### Spring/Stator Contact Control Point (if Pivoting Type)

The name of the control point representing the contact point between the spring and stator. This point should be placed at the contact point on the spring centerline axis. This attribute applies only to pivoting style pumps, and should be set to "ign" for sliding style pumps. See the schematic below illustrating this location.

### Spring End Control Point (if Pivoting Type)

The name of the control point representing the endpoint of the spring, and it should be placed at the point on the spring centerline axis. This attribute applies only to pivoting style pumps, and should be set to "ign" for sliding style pumps. See the schematic below illustrating this location.

### Spring Preload

Preload of the spring. This should normally be a positive value representing the spring in compression.

### Spring Stiffness

Stiffness of the spring.

### Control Chamber 1 Start Control Point

The name of the control point representing the starting point of the first control chamber. See the schematic below illustrating this location.

### Control Chamber 1 End Control Point

The name of the control point representing the ending point of the first control chamber. See the schematic below illustrating this location.

### Control Chamber 2 Start Control Point

The name of the control point representing the starting point of the second control chamber. See the schematic below illustrating this location.

### Control Chamber 2 End Control Point

The name of the control point representing the ending point of the second control chamber. See the schematic below illustrating this location.





## Options

### **Housing Leakage Gap Between Vane Pockets**

The leakage gap between the pump housing and the side of the vane. This should represent the gap of only one side, as the vane is assumed to be symmetrically placed between the two sides of the housing. A flow rate multiplier of 2 will be applied to represent leakage from both sides of the vane in the 'LeakageRectangular' connection.

### **Housing Leakage Gap From Outlet to Inlet Volume**

The leakage gap from the outlet volume to the inlet volume, representing the gap between the pump housing and the rotor. This should represent the gap of only one side, as the rotor and vanes are assumed to be symmetrically placed between the two sides of the housing. A flow rate multiplier of 2 will be applied to represent leakage from both sides of the vane in the 'LeakageRectangular' connection.

### **Port Area Multiplier for Inlet 1**

Multiplier to the port area for inlet 1. Typically this multiplier will be set to 2 when the user wishes to model an inlet on both sides of the pump with a single volume.

### **Port Area Multiplier for Outlet 1**

Multiplier to the port area for outlet 1. Typically this multiplier will be set to 2 when the user wishes to model an outlet on both sides of the pump with a single volume.

### **Port Area Multiplier for Inlet 2**

Multiplier to the port area for inlet 2. Typically this multiplier will be set to 2 when the user wishes to model an inlet groove on both sides of the pump with a single volume.

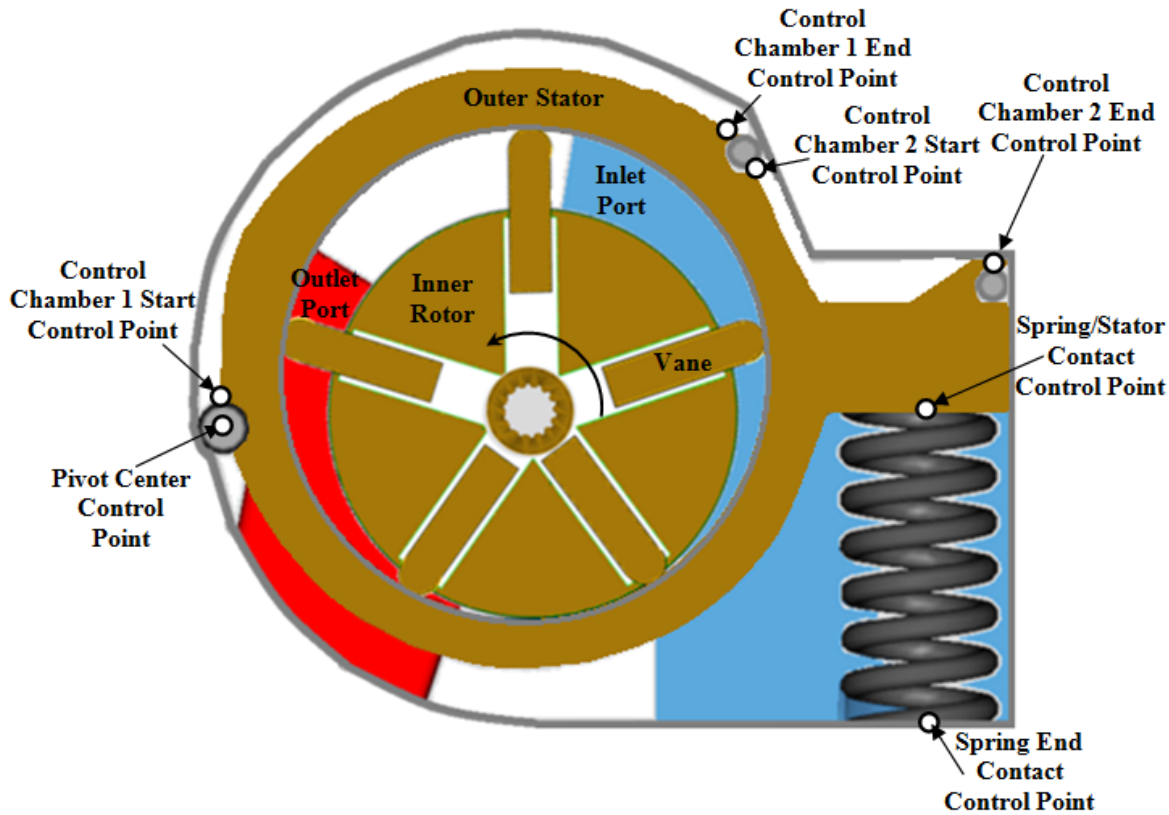
### **Port Area Multiplier for Outlet 2**

Multiplier to the port area for outlet 2. Typically this multiplier will be set to 2 when the user wishes to model an outlet groove on both sides of the pump with a single volume.

### **Vane pump geometry**

The schematic below shows the geometry of the vane pump and the locations of the control points required to create a mechanical assembly for dynamic eccentricity prediction.





### How to create a control point

To create a control point, the user can either create a "Global" control point or "Component" control point. To create a Global control point, go to the Dimensions Menu -> Create Global Control Point ..., then type in the X,Y,Z coordinates representing the point.

Often the user does not know the X,Y,Z coordinates directly, so it is much easier to use the Dimensions Menu -> Create Component Control Point ... option, where the user simply selects a point on a given mesh shape. Note a control point cannot be used on a solid shape at this time. Also note if the mesh shape is translated or rotated in the model after making a control point on the shape, the global coordinates of the control point will move with the shape. The user can optionally convert a Component Control Point to a Global control point to prevent the point from moving with the shape. This is done by selecting the Component control point and right-clicking and selecting "Convert to Global Control Point".





## **GEMWoolAll - Wool Inside a Sleeve or Flowsplit**

This template is used to model sound absorbing material such as wool inside a sleeve or flowsplit. The entire component will be filled with the absorbing material.

### **Main**

---

#### **Wool Material**

Name of a 'Wool' reference object that defines the properties of the absorbing material in the sleeve or flowsplit.







## **GEMWoolChamber - Wool Inside Shell Chamber**

This template is used to model sound absorbing material such as wool inside a chamber of a shell. The entire chamber will be filled with the material.

### **Main**

---

#### **Wool Material**

Name of a 'Wool' reference object that defines the properties of the absorbing material in the sleeve.

#### **Thickness of Wool Layer**

Thickness of the wool layer attached to the inside surface of the shell in the chamber. A very large thickness here would be the equivalent of the entire chamber being filled with wool. A value of "def" specifies that wool will be added to the entire chamber. The baffles that confine the chamber are also assumed to have a wool layer.





## GEMWoolShell - Wool Inside Shell

This template is used to model sound absorbing material such as wool in a shell.

### Main

<b>Wool Material</b>	Name of a 'Wool' reference object that defines the properties of the absorbing material in the shell.
<b>Thickness of Wool Layer</b>	Thickness of the wool layer attached to the inside surface of the shell. A very large thickness here would be the equivalent of the entire shell being filled with wool. A value of "def" specifies that wool will be added to the entire shell.
<b>Include End Baffles</b>	Flag to add wool to the end baffles of the shell. If a <b>Thickness of Wool Layer</b> is specified, this option will determine if that wool layer is added to the end baffles or not (just the outer wall). If the <b>Thickness of Wool Layer</b> attribute above is "def" such that the entire shell contains wool, then this attribute doesn't do anything.

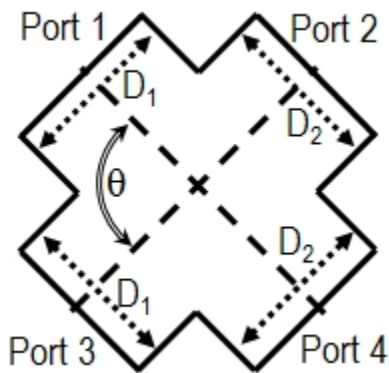


## GEMXsplit - X-Shaped Flowsplit

This template is used to describe a flowsplit volume connected to 4 flow components in an X-configuration.

### Geometry

Diameter of Port 1	Diameter of the first 2 ports (ports 1 and 3 in the diagram below).
Diameter of Port 2	Diameter of the next 2 ports opposite the first ones (ports 2 and 4 in the diagram below). A value of "def" will use a diameter equal to the <b>Diameter of Port 1</b> .
Crossing Angle ( $\theta$ )	Angle that the flow paths cross. This is $\theta$ in the diagram below and essentially gives the angle between ports 1 and 3 in the diagram (and between ports 2 and 4 in the diagram). A value of "def" will use 90 degrees.
Wall Thickness	Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a 'WallThermalProperty' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the 'WallThermalProperty' reference object.
Volume	Specifies the volume of the component. Setting this attribute to "def" will use the actual volume of the component as calculated by GEM3D.



Schematic Representation of the X Shaped Flowsplit Template

### Location

Location X	Specifies the absolute X location of the component's centroid in the <a href="#">Global Coordinate System</a> .
Location Y	Specifies the absolute Y location of the component's centroid in the <a href="#">Global Coordinate System</a> .
Location Z	Specifies the absolute Z location of the component's centroid in the <a href="#">Global Coordinate System</a> .





<b>Direction X</b>	Specifies the X component of the vector describing the direction that the component will be created. Only a unit vector is needed to describe the dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Y</b>	Specifies the Y component of the vector describing the direction that the component will be created. Only a unit vector is needed to describe the dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
<b>Direction Z</b>	Specifies the Z component of the vector describing the direction that the component will be created. Only a unit vector is needed to describe the dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.

## Visual

<b>Transparency Percent</b>	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
<b>Display Color</b>	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> <li>• Black</li> </ul>

## Main

### Basic Geometry and Initial Conditions

#### Surface Area

Specify required geometric input as well as the initial state of the fluid in the flow component.

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**





	<p><math>D</math> = <b>Diameter</b> (specified above)</p> <p><math>L</math> = <b>Length</b> (specified above)</p> <p><math>D_{orifice}</math> = orifice diameters adjacent to each port</p>
<b>Initial State Name</b>	Name of the 'FluidInitialState' reference object describing the initial conditions inside the pipe.
<b>Surface Finish</b>	Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.
<p>☉ <b>Smooth</b></p> <p>☉ <b>Roughness from Material</b></p>	<p>Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).</p> <p>This attribute gives choices of materials that may be used to specify the surface roughness.</p> <p><u>Material Name : Default Roughness Value (mm)</u></p> <ul style="list-style-type: none"> <li>• <b>drawn_metal</b> : 0.002</li> <li>• <b>steel</b> : 0.046</li> <li>• <b>cast_iron</b> : 0.26</li> <li>• <b>light_rust_steel</b> : 0.25</li> <li>• <b>heavy_rust_steel</b> : 1.0</li> <li>• <b>smooth_plastic</b> : 0.0025</li> <li>• <b>smooth_rubber</b> : 0.025</li> <li>• <b>smooth_galvanized</b> : 0.025</li> <li>• <b>normal_galvanized</b> : 0.15</li> <li>• <b>wrought_iron</b> : 0.046</li> <li>• <b>asphalted_cast</b> : 0.12</li> <li>• <b>extruded_aluminum</b> : 0.003</li> <li>• <b>user_value</b>: 0.0</li> </ul>
☉ <b>Sand Roughness</b>	<p>Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.</p> <p>A numeric value or a parameter may be entered for this attribute.</p>
<b>Additional Geometry Options</b>	Specify optional geometric characteristics of the flow component.
<b>No. of Identical Pipes</b>	Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)





## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

#### ☉ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall temperature distribution, where X is normalized length with range 0 to 1.

#### ☉ Calculated Wall Temperature

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

#### ☉ Wall Layer Properties Object

Name of the 'WallThermalProperty' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

#### ☉ Wall External Boundary Conditions Object

Name of the 'WallThermalBoundary' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

#### ☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

#### ☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('ConvectionConn') to a thermal primitive part (i.e. 'ThermalMass'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

#### ☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

### Additional Thermal Options

Specify optional thermal characteristics of the flow component.





<b>Heat Transfer Multiplier</b>	Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for <b>Surface Finish</b> , additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")
<b>Heat Input Rate</b>	The rate of heat input to the fluid or the name of a dependency reference object.
<b>Thermocouple Object</b>	Name of a 'Thermocouple' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe* or FlowSplit* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.
<input checked="" type="radio"/> <b>Heat Transfer Correlation (Colburn)</b>	Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.
<input checked="" type="radio"/> <b>User Defined Heat Transfer Model</b>	Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the 'UserModel' object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the <b>Heat Transfer Multiplier</b> value.
<input checked="" type="radio"/> <b>Heat Transfer Coefficient</b>	Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.
<b>Condense/Evaporate Water Vapor (Non-Refrigerant Circuits)</b>	<p>Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to 'FluidRefrigerant' Circuits, which do not require any settings to enable boiling/condensation.</p> <ul style="list-style-type: none"> <li>• <b>off</b>: No condensation or evaporation is modeled.</li> <li>• <b>on_gas</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.</li> <li>• <b>on_wall</b>: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.</li> </ul> <p>Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as <b>off</b> and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These</p>





RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an '[EjectorConn](#)' object if desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity '[SensorConn](#)' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

### Flowsplit HTC Calculation Method

#### Method #1 (v7.5 and prior)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, and then they are weighted by the boundary area. This method tends to under predict the heat transfer coefficient in the flowsplit, and is only recommended to reproduce previous version results.

#### Method #2 (Recommended)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port. The resulting coefficients are averaged based on the boundary flow rates. This method produces a more accurate prediction for the heat transfer coefficient, and is recommended especially in applications where the heat transfer in the flowsplit is a significant part of the overall circuit heat transfer rate.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify







his/her own model using **User Defined Friction Model**.

☉ **Friction Multiplier**

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

☉ **No Friction Pressure Losses**

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

**User Defined Friction Model**

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction Multiplier** value.





## GEMYsplit - Y-Shaped Flowsplit

This template is used to describe a flowsplit volume connected to three flow components in a Y-configuration.

In most cases this component is discretized into a general flowsplit ('[FlowSplitGeneral](#)') part. However, in the particular situations given below the geometry of the component is such that it can be discretized into more specialized components.

### Symmetric Y flowsplit

If the following 3 geometry conditions are met, then this component will be discretized into a Y-shaped flowsplit ('[FlowSplitY](#)') part.

- 1) **Angle of Port 2** must be equal to **Angle of Port 3**
- 2) **Angle of Port 2** must be between 105 deg and 180 deg (or equal to 105 or 180)
- 3) The lengths of all 3 ports must be equal

In this situation, the attributes **Flow Characteristic (Phasing)**, **Total Y Branch Expansion Area**, and **Out-of-Phase Expansion Factor** will be used. Since they are only applicable to the '[FlowSplitY](#)' template, they will be ignored in any other situation.

### T flowsplit

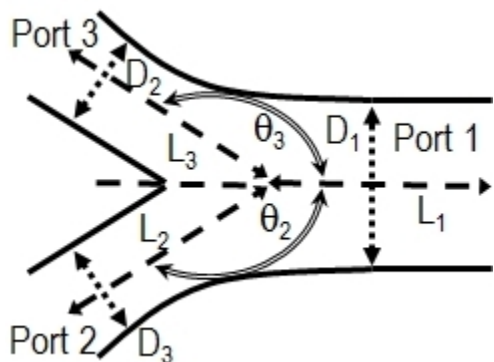
If the following 4 geometry conditions are met, then this component will be discretized into a T-shaped flowsplit ('[FlowSplitTAngle](#)') part.

- 1) All 3 diameters must be equal
- 2) **Angle of Port 2** or **Angle of Port 3** must equal 180 deg
- 3) The angle that is not 180 deg must be between 45 and 135 deg (or equal to 45 or 135)
- 4) The length of the branch port (the port that is not 180 deg) must be less than the sum of the other 2 ports

## Geometry

<b>Diameter of Port 1</b>	Diameter of the main port (port 1).
<b>Length of Port 1</b>	Length of the section of port 1.
<b>Diameter of Port 2</b>	Diameter of the second port (port 2).
<b>Length of Port 2</b>	Length of the section of port 2.
<b>Angle of Port 2</b>	Angle between port 2 and port 1. It is $\theta_2$ in the schematic below.
<b>Diameter of Port 3</b>	Diameter of the third port (port 3).
<b>Length of Port 3</b>	Length of the section of port 3.
<b>Angle of Port 3</b>	Angle between port 3 and port 1. It is $\theta_3$ in the schematic below.
<b>Wall Thickness</b>	Wall thickness of the component. This will be used to draw the component in the graphical window. The wall thickness used in the thermal wall solver in GT-SUITE must be defined in a ' <a href="#">WallThermalProperty</a> ' reference object. If "def" is entered, then the thickness that is displayed will be the total thickness entered in the ' <a href="#">WallThermalProperty</a> ' reference object.





Schematic Representation of the 'GEMYsplit' Template

Location

Location X	Specifies the absolute X location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
Location Y	Specifies the absolute Y location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
Location Z	Specifies the absolute Z location of the component's first cross section in the <a href="#">Global Coordinate System</a> .
Direction X	Specifies the X component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
Direction Y	Specifies the Y component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.
Direction Z	Specifies the Z component of the vector describing the direction that the component will be extruded. Only a unit vector is needed to describe the extrusion dimension, so the <b>Direction X</b> , <b>Direction Y</b> , and <b>Direction Z</b> attributes may be replaced with the equivalent unit vector.

Visual

Transparency Percent	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent. A value of "def" will use the value specified with the <b>Default Transparency Percentage</b> option in File → Options → General.
Display Color	Indicates the color used when drawing the part. The color choices include: <ul style="list-style-type: none"><li>• Red</li><li>• Blue</li></ul>





- Dark Blue
- Green
- Copper
- Gold
- Grey
- Black

## Main

### Basic Geometry and Initial Conditions

Specify required geometric input as well as the initial state of the fluid in the flow component.

### Flow Characteristic (Phasing)

There are two options for the flow characteristic:

- **In-Phase** indicates that the flow at ports 2 and 3 are simultaneous.
- **Out-of-Phase** indicates that the flow at ports 2 and 3 are not simultaneous.

For further information about In-Phase and Out-of-Phase flow configurations, please see the context help for the ['FlowSplitY'](#) template.

*Note: This attribute is only used when this component is discretized into a ['FlowSplitY'](#) part. See above comments containing criteria for this situation.*

### Surface Area

The surface area used in both heat transfer and friction calculations for the flowsplit. If "def" is entered for this attribute, the surface area will be estimated by assuming that the flowsplit is cylindrical in shape and then subtracting the area of the flowsplit openings with the following formula:

$$Area = \pi D * L + 2 \frac{\pi}{4} D^2 - \sum_1^3 \frac{\pi}{4} D_{orifice}^2$$

where:

$Area$  = **Surface Area**

$D$  = **Diameter** (specified above)

$L$  = **Length** (specified above)

$D_{orifice}$  = orifice diameters adjacent to each port

### Initial State Name

Name of the ['FluidInitialState'](#) reference object describing the initial conditions inside the pipe.

### Surface Finish

Specify surface finish characteristics of the inside of the wall in contact with the fluid. This value will influence the pressure drop through this component, but in many situations it has a more significant effect on the heat transfer coefficient, and therefore the fluid temperature.

### ☉ Smooth

Corresponds to a roughness of 0.0 mm. This does not mean that there are no frictional losses, but that the frictional losses will be that of a perfectly smooth surface (like glass).





## ◎ Roughness from Material

This attribute gives choices of materials that may be used to specify the surface roughness.

Material Name : Default Roughness Value (mm)

- **drawn\_metal** : 0.002
- **steel** : 0.046
- **cast\_iron** : 0.26
- **light\_rust\_steel** : 0.25
- **heavy\_rust\_steel** : 1.0
- **smooth\_plastic** : 0.0025
- **smooth\_rubber** : 0.025
- **smooth\_galvanized** : 0.025
- **normal\_galvanized** : 0.15
- **wrought\_iron** : 0.046
- **asphalted\_cast** : 0.12
- **extruded\_aluminum** : 0.003
- **user\_value**: 0.0

## ◎ Sand Roughness

Surface roughness value using the "Sand Roughness" scale (Sand Roughness divided by the diameter of the pipe is "Relative Roughness", the friction factor that is used in the Moody Diagram). A table of sand roughness values for common materials is given in the User's Manual in the chapter on Pipes under the subheading Friction Losses.

A numeric value or a parameter may be entered for this attribute.

## Additional Geometry Options

Specify optional geometric characteristics of the flow component.

## No. of Identical Pipes

Number of identical, parallel pipes represented by this pipe (usually 1.0, except for modeling many small pipes inside some heat-exchangers). ("def"=1)

## Thermal

### Wall Temperature Method

Specify the thermal characteristics of the flow component. In many cases, the **Calculated Wall Temperature** option will give the most realistic results. For well-insulated pipes or isothermal studies, the **Adiabatic** option should be used, which will ignore heat losses to ambient.

## ◎ Imposed Wall Temperature

Impose a temperature which represents the temperature of the wall surface directly in contact with the fluid. The heat transfer rate between the wall and fluid will be calculated using this imposed wall temperature, along with the the actual local fluid temperature and heat transfer coefficient in each subvolume.

A wall temperature value or the name of a dependency reference object can be specified here. 'XYTable' can be used to impose an axial wall





☉ Calculated Wall Temperature

temperature distribution, where X is normalized length with range 0 to 1.

Activates the built-in thermal wall solver to solve for the temperature of the walls and the resulting heat transfer rate to the fluid, accounting for wall material properties and ambient conditions external to the wall (i.e. external free-flow temperature and heat transfer coefficient). The wall temperature will be calculated based on the inputs for the attributes below, which allow for modeling multiple layers within the wall material.

☉ Wall Layer Properties Object

Name of the '[WallThermalProperty](#)' reference object defining the thermal properties of the wall (i.e. thickness, material, and emissivity).

☉ Wall External Boundary Conditions Object

Name of the '[WallThermalBoundary](#)' reference object defining the boundary conditions of the wall (i.e. ambient temperature and external convection coefficient).

☉ Initial Wall Temperature

Temperature of the wall at the first time step of the simulation.

☉ Wall Temperature from Connected Thermal Primitive

Obtains the wall temperature used for calculation of the heat transfer rate to the internal fluid from an externally connected thermal primitive part. The flow part must be connected via a thermal connection ('[ConvectionConn](#)') to a thermal primitive part (i.e. '[ThermalMass](#)'). This option is useful when both fluid sides of the wall are modeled as flow circuits, or when the wall will be connected thermally to other parts in the model (i.e. conduction heat transfer to other masses).

☉ Adiabatic

Imposes a zero heat transfer rate between the wall and the fluid. If chosen, any non-zero value for the **Heat Transfer Multiplier** will be ignored. However, a non-zero **Heat Input Rate** may still be entered and this rate will be imposed on the fluid (this is a separate "source term", not tied to the wall heat transfer rate).

Additional Thermal Options

Specify optional thermal characteristics of the flow component.

Heat Transfer Multiplier

Heat transfer coefficient multiplier. The calculated heat transfer rate between the fluid and the wall will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional heat transfer over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Heat Transfer Enhancement Factor")

Heat Input Rate

The rate of heat input to the fluid or the name of a dependency reference object.

Thermocouple Object

Name of a '[Thermocouple](#)' reference object that defines the properties of the absorbing material in the flowsplit. This reference object models a Thermocouple immersed in a fluid in a Pipe\* or FlowSplit\* part. The Thermocouple temperature is predicted based on convective heat transfer to the fluid and radiation heat transfer to the pipe wall.





☉ Heat Transfer  
Correlation (Colburn)

Select this option to use the Colburn heat transfer correlation to determine the heat transfer coefficient for all parts in the flow system. This is the standard option for the GT flow solver.

☉ User Defined Heat  
Transfer Model

Select this option to define a custom heat transfer model to take the place of the standard Colburn solution in the GT solver. The attribute should be used to define the name of the ['UserModel'](#) object which will be used to calculate the Heat Transfer Coefficient (between the fluid and the wall) value. The heat transfer coefficient value that is calculated through the user model will also be multiplied by the **Heat Transfer Multiplier** value.

☉ Heat Transfer  
Coefficient

Select this option to directly impose the Heat Transfer Coefficient (between the fluid and the wall) value.

Condense/Evaporate  
Water Vapor (Non-  
Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water on the basis of the relative humidity of the mixture. This attribute does not apply to ['FluidRefrigerant'](#) Circuits, which do not require any settings to enable boiling/condensation.

- **off**: No condensation or evaporation is modeled.
- **on\_gas**: Condensation and evaporation is enabled and the latent heat is released entirely to the fluid.
- **on\_wall**: Condensation and evaporation is enabled and the latent heat is released entirely to the wall.

Enabling water condensation and evaporation modeling is the most rigorous but in general also the most computationally expensive option for modeling systems in which condensation may occur. In many cases, it is not necessary to literally model condensation phenomena because the mere occurrence of condensate is problematic and not tolerated (typically for durability reasons). If with this constraint, it is recommended to leave this option as **off** and instead monitor the "Water Saturation (Dew) Temperature" and "Minimum Temperature Above Water Saturation Temperature" variables in flow components. These RLT variables must be activated by the user in Output Setup\OtherOutput. Be aware that although cheaper than modeling the condensation/evaporation process, these optional RLTs are also associated with a noticeable computational cost. If the "Minimum Temperature Above Water Saturation Temperature" goes negative, you can conclude that the mixture has become overcooled.

If this flag is set to **on\_gas** or **on\_wall** the properties of both vapor and liquid water must be defined.

Whenever possible, it is recommended to only have a single h2o vapor and single h2o liquid species in the project tree to prevent confusion. In the event multiple liquid h2o species or multiple vapor h2o species are defined in the project tree, the solver will print a warning message displaying which species is the selected condensate/vaporizing species.

Condensed water can be withdrawn using an ['EjectorConn'](#) object if





desired. This feature is often helpful in the modeling of EGR coolers.

NOTE: Although implemented for both the explicit and implicit solvers, use of the implicit humidity solver may result in inaccuracies in the condensation and evaporation calculations when using sufficiently large timesteps and modeling high heat transfer rates. For these models mixture relative humidities may depart from the expected relative humidity, which can be observed using the Relative Humidity 'SensorConn' output signal. If a study is particularly sensitive to humidity calculations and this inaccuracy of the implicit solver is unsuitable, improved condensation and evaporation accuracy can be obtained by using the explicit solver.

### Flowsplit HTC Calculation Method

#### Method #1 (v7.5 and prior)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port, and then they are weighted by the boundary area. This method tends to under predict the heat transfer coefficient in the flowsplit, and is only recommended to reproduce previous version results.

#### Method #2 (Recommended)

This method of calculating the heat transfer coefficient in a flowsplit uses the boundary velocities to calculate a heat transfer coefficient for each port. The resulting coefficients are averaged based on the boundary flow rates. This method produces a more accurate prediction for the heat transfer coefficient, and is recommended especially in applications where the heat transfer in the flowsplit is a significant part of the overall circuit heat transfer rate.

## Pressure Drop

### Friction Options

Specify whether the pressure losses due to friction between the fluid and interior wall of the flow component should be accounted for. A user can use the built in model for calculating the friction coefficient or specify his/her own model using **User Defined Friction Model**.

#### ☐ Friction Multiplier

Friction coefficient multiplier. The calculated pressure loss due to friction between the fluid and the interior wall surface will be scaled by this factor. (If a non-smooth option is selected for **Surface Finish**, additional pressure loss over a smooth pipe will already be calculated, so this attribute should typically be set to 1.0.) ("def"=1.0) (Also see FlowControl folder of Run Setup for information about the global "Friction Enhancement Factor")

#### ☐ No Friction Pressure Losses

Pressure losses due to surface friction effects are neglected. This is equivalent to specifying the **Friction Multiplier** as zero.

#### User Defined Friction Model

The name of the 'UserModel' object which will be used to calculate the Friction Coefficient value. If a user model is not going to be used this attribute should be set to "ign". The friction coefficient value that is calculated through the user model will also be multiplied by the **Friction**







**Multiplier** value.





## GerotorPump - Gerotor Pump Model Generator

This template is used to generate a predictive gerotor pump flow model from an existing CAD model of a gerotor pump. In order to use this template, a CAD model must be supplied for the outer gear, inner gear, inlet port volume and outlet port volume of the gerotor pump. By identifying these four parts, GEM will be able to calculate the volume, inlet area and outlet area profiles of the pump, and assemble the model with the correct configuration of parts.

An example model can be found under:

GTIHOME\X.X.X\examples\GEM3D\Lubrication\GerotorPumpExample.gem

### Main

<b>Inner Gear Mesh Shape</b>	Name of the mesh shape that represents the inner gear. It is recommended to first select the mesh shape on the map representing the inner gear, give it a descriptive name, then right-click in this attribute cell and point to the name of the mesh shape.
<b>Outer Gear Mesh Shape</b>	Name of the mesh shape that represents the outer gear. It is recommended to first select the mesh shape on the map representing the outer gear, give it a descriptive name, then right-click in this attribute cell and point to the name of the mesh shape.
<b>Inlet Volume Mesh Shape</b>	Name of the mesh shape that represents the inlet volume. It is recommended to first select the mesh shape on the map representing the inlet volume, give it a descriptive name, then right-click in this attribute cell and point to the name of the mesh shape.
<b>Inlet 2 Volume Mesh Shape</b>	Name of the mesh shape that represents the second inlet volume, if one exists. If a second inlet volume does not exist, then this attribute can be set to "ign". It is recommended to first select the mesh shape on the map representing the second inlet volume, give it a descriptive name, then right-click in this attribute cell and point to the name of the mesh shape.
<b>Outlet Volume Mesh Shape</b>	Name of the mesh shape that represents the outlet volume. It is recommended to first select the mesh shape on the map representing the outlet volume, give it a descriptive name, then right-click in this attribute cell and point to the name of the mesh shape.
<b>Outlet 2 Volume Mesh Shape</b>	Name of the mesh shape that represents the second outlet volume, if one exists. If a second outlet volume does not exist, then this attribute can be set to "ign". It is recommended to first select the mesh shape on the map representing the second outlet volume, give it a descriptive name, then right-click in this attribute cell and point to the name of the mesh shape.
<b>Angle Increment for Output Volume / Area Profiles</b>	<p>Specifies the resolution of the output volume, inlet area, and outlet area profiles, to the nearest degree. Typically, resolution of 1° is recommended, as this resolution is required to capture the proper opening and closing points for the inlet and outlet area profiles.</p> <p>Note the output profiles will often not produce the exact increment prescribed in this attribute, as GEM internally converts this angle attribute to a number of "snapshots", or times to rotate the gears of the</p>





pump, where the number of snapshots, S, is calculated using the equation below.

$$S = 360/RN$$

where R represents the value used in this attribute, and N represents the number of outer gear teeth. As an example, a gerotor pump with 5 outer teeth and angle increment set to 1 degree will take a total of 72 snapshots, which means GEM will rotate the pump 72 times (in 1 degree increments) and record the volume, inlet area and outlet area for each of the 5 voids between the inner and outer gear, giving a total of 360 data points in the profiles.

**Include Friction Model?**

If this checkbox is on, then a friction component will be added to the gerotor pump assembly with most input dimensions automatically calculated. There are a few inputs requiring manual input from the user including various clearances and support bearing dimensions. If this checkbox is off, friction will be ignored.



## LocalOrigin - Local Coordinate System

This template is used to create a local coordinate system which can be used to define the position of reference/additional nodes in 'GEMInertia3D' components. Alternatively, the 'LocalOrigin' components are automatically imported along with the 3D solid CAD model with pre-filled attributes.

### Main

<b>Location X (Global)</b>	Initial X- position of the 'LocalOrigin' with respect to the CAD Global (World) Coordinate System
<b>Location Y (Global)</b>	Initial Y- position of the 'LocalOrigin' with respect to the CAD Global (World) Coordinate System
<b>Location Z (Global)</b>	Initial Z- position of the 'LocalOrigin' with respect to the CAD Global (World) Coordinate System
<b>1<sup>st</sup> Euler Angle (about Global Z)</b>	First Euler angle (rotation around the 'CAD Global (World) Coordinate System' Z- axis) used to specify the initial orientation of the 'LocalOrigin' with respect to the 'CAD Global (World) Coordinate System' Z- axis ("def"=0.0)
<b>2<sup>nd</sup> Euler Angle (about Local X)</b>	Second Euler angle (rotation around the 1 <sup>st</sup> intermediate frame) used to specify the initial orientation of the 'LocalOrigin' with respect to the 'CAD Global (World) Coordinate System' Z- axis ("def"=0.0)
<b>3<sup>rd</sup> Euler Angle (about Local Z)</b>	Third Euler angle (rotation around the 2 <sup>nd</sup> intermediate frame) used to specify the initial orientation of the 'LocalOrigin' with respect to the 'CAD Global (World) Coordinate System' Z- axis ("def"=0.0)

### Visual

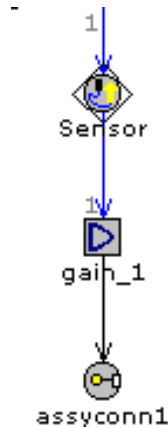
<b>Axis Length</b>	Defines the length of the three (XYZ) axes that will be displayed in the graphical window at the location and orientation specified by the user.
<b>Transparency Percent</b>	Indicates the transparency level used when displaying the local coordinate system. 0 indicates opaque (solid) and 90 indicates almost completely transparent.





## SensorConn3D - Sensor Connection

This template is used to sense quantities (pressure, temperature, velocity, etc) in a flow part to pass the value to a control components part in the discretized .gtm model. The discretized model will have a 'SensorConn' and 'Gain' setup to automatically connect to other control components in the model.



### Main

#### Subassembly Port Number

The number entered here will be used as the port number for the 'SensorConn'. If "def" is entered a number will be automatically assigned.

#### Signal Quantity to Sense

Access the Value Selector to select the sensed quantity for the object.

(only the attributes associated with the selected radio button need to be filled out)

#### Positioning for FlowSplit\* or Pipe\*

##### ☉Coordinate System

Specifies the reference location of the sensor in relation to the parent object or global origin:

- **local** indicates that the location entered below will be measured from the local origin of the parent object.
- **global** indicates that the location entered below will be measured from the global origin.

##### (☉) Location X

Specifies the absolute X location of the component in the local or [Global Coordinate System](#).

##### (☉) Location Y

Specifies the absolute Y location of the component in the local or [Global Coordinate System](#).

##### (☉) Location Z

Specifies the absolute Z location of the component in the local or [Global Coordinate System](#).

##### ☉Normalized Sensor Location for Pipe\* Parts

**For Pipe\* Parts:** This attribute is used to specify a normalized axial location in 'Pipe\*' components at which the desired quantity is to be sensed (because there may be several discretized volumes within





each 'Pipe\*' part). The normalized location is a value between 0 and 1 with "0" defining the end of the pipe at port 1 (the inlet) and "1" defining the end of the pipe at port 2 (the outlet). The **Interpolation Option** attribute below determines exactly how this location will be used.

When sensing from 'Pipe\*' components, setting this attribute to "ign" will cause the selected quantity to be sensed at a normalized location of 0.

### (☉) Interpolation Option for Pipe\* Parts

One of the following choices:

- **use\_nearest\_volume** indicates the sensor will report the value in the nearest subvolume to the specified location. For example, if a given 'Pipe\*' part has two sub-volumes, all specified sensor locations between 0.0 and 0.5 will report the value from the first subvolume.
- **interpolate** indicates that the sensor will interpolate between the nearest two subvolumes. If locations are requested outside of the outer subvolume centroid, the centroid value will be reported. For example, if a given 'Pipe\*' part has two sub-volumes (i.e. centroid locations are at 0.25 and 0.75): requests between 0.0 and 0.25 will report the value from the first subvolume, request between 0.25 and 0.75 will be interpolated between the two subvolume values, and values between 0.75 and 1.0 will report the value from the second subvolume.

## Advanced

The three attributes below are used when sensing "Species Mass Fraction" or "Species Mole Fraction" from flow components, or "Species Mass Flow Rate" or "Species Molar Flow Rate" from flow connections. One of the three radio buttons below must be selected.

### ☉ Standard Sensed Species Name

Any of the 13 standard species: **CO2, H2O, N2, O2, CO, H2, H, O, OH, NO, N, AR, SO2**. Note this option is required for sensing 'FluidPredfined' prod\_\* fluid objects. For all applications involving a cylinder or aftertreatment component this option should be used to sense species with the molecular formulas mentioned above.

### (☉) Standard Sensed Species Specification

This attribute is primarily for studying EGR or O2 pass through in detail. For most applications "total" is the safest and recommended option.:

- **unburned**: This option includes only the "unburned" species that is present, for example O2 from fresh air in the intake manifold would be flagged as "unburned". Also if that O2 passes through the cylinder during valve overlap, but prior to combustion start, there will be "unburned" O2 in the exhaust system.
- **burned**: This option includes only the "burned" species that is present. Any of the 13 standard species produced by a combustion or kinetic reaction object will be internally flagged as "burned". Also any of the 13 standard species that passes through a cylinder or reactor while combustion/reaction is occurring, even if that particular species is unreacted, will be flagged as "burned". For example excess O2 from lean combustion will be flagged as "burned", therefore there will be "burned" O2 in the exhaust system.





	<ul style="list-style-type: none"> <li>• <b>total:</b> This option senses the combination of a basic combustion product/burned species and the basic unburned species. For example if a user wishes to sense the "total" O2 concentration, this option will include both the "burned" and "unburned" O2 that is present.</li> </ul>
<b>©User-Defined Sensed Species Object</b>	The name of the ' <b>Fluid*</b> ' reference object that is being sensed. If there is no cylinder or aftertreatment component in the model, and the species being sensed is one of the 13 "standard" species as mentioned above, then this attribute should be used, because the "standard" species won't be created (in order to reduce computation time). Value Selector may be used to select the fluid reference object from the template library. This attribute radio button should be selected and value set to "ign" when not sensing a species.
<b>©Coverage or Combined Mixture Vapor Species Name</b>	<p>The text name of a surface coverage species used when sensing "Average Coverage" from a '<b>ChemConn</b>' between a '<b>SurfaceReactions</b>' part and an aftertreatment flow component. For example "Ce2O3". The name must be typed as text.</p> <p>This attribute has the dual purpose of also supporting the sensing of combined mixture vapor species, the vapor of a '<b>FluidMixtureCombined</b>'. For example if there is a combined mixture named "E85" in the model, then "E85-vap" may be typed here to sense the combined mixture vapor that is created internally by the solver. Again the sensor link must be set to Species Mass or Mole Fraction or Species Mass or Molar Flow Rate.</p>
<b>Calculation for Sensed Quantity for Matrix* Part</b>	<p>The selected choice will determine the mathematical operation to be performed on the sensed quantity from the '<b>Matrix*</b>' part. This option can only be used in COOL3D models.</p> <ul style="list-style-type: none"> <li>• <b>average:</b> If "average" is selected, then "pressure" is averaged over the total volume, "velocity" is averaged over the area, and any other quantity is simply averaged over the number of subvolumes that exist in the part.</li> <li>• <b>local:</b> If "local" is selected, then the quantity will be sensed from the local element found at the location entered in the <b>Main</b> folder.</li> </ul>

## Visual

<b>Display Color</b>	<p>Indicates the color used when drawing the sensor. The color choices include:</p> <ul style="list-style-type: none"> <li>• Red</li> <li>• Blue</li> <li>• Dark Blue</li> <li>• Green</li> <li>• Copper</li> <li>• Gold</li> <li>• Grey</li> </ul>
----------------------	---





- **Black**

## Show Sensor Label

If checked, this will display the sensor name on the canvas.

## Averaging and Smoothing

This folder offers options for averaging and smoothing signals. When activated, the functions are identical to those available through the 'MovingAverage' and 'FirstOrderFilter' templates, but it is often more convenient to use the options right from the sensor's signal selection dialog box rather than putting a separate part on the map. If the plot request for the sensor is on and averaging/smoothing is on, both the raw incoming signal and averaged/smoothed outgoing signals will be saved in the plot.

### ☐ No Averaging or Smoothing (Directly Use Sensed Signal)

Averaging and smoothing options are not active.

### ⊙ MovingAverage: Number of Cycles

Moving average smoothing with the window width specified in number of cycle. Non-integer values are allowed. This option is only allowed when a simulation is configured in Run Setup as "Periodic". The "Main Driver" specified in the Time Control folder of Run Setup provides the angle that is used for calculating the period.

### (⊙) Maximal Duration of Window (for Low Frequencies)

Optional maximum time limit on the averaging window duration. This option is typically used when running transient simulations where the main driver (for example, the engine crankshaft) goes to very low or even zero speed (for example, if the engine crankshaft is temporarily shut off during start-stop operation). This attribute may be set to "ign" if no limit on the window duration is necessary.

### ⊙ MovingAverage: Time Duration

Moving average smoothing with the window width specified as a time duration.

### ⊙ FirstOrderFilter: Time Constant

The time constant of the filter. The value may be a constant or the name of an 'RLTDependence\*' reference object.

The use of the time constant is defined by the following equations, presented both as a Laplace transform and as a differential equation:

$$Y(s) = \frac{1}{\tau s + 1} U(s) \quad \tau \dot{y} = -y + u$$

where:

$\tau$  = **Time Constant**

$y$  = "Output" signal

$u$  = "Input" signal

$s$  = Laplace complex variable

## Initial Output

Value of the output signal at the start of the simulation. If Initialization State of the simulation is **previous\_case** (see Initialization folder in RunSetup), then this attribute will only be used for the first case, and the initial output for subsequent cases will be taken from the end of the







previous case. If this attribute is set to "def", the initial output will be the initial value of the input signal (value at the 0th time step).



**ThermalBound – Convection Boundary Condition**

This reference object is used to apply a thermal boundary condition to a thermal mass or finite element. When this reference object is used, the exported model will contain a ‘Temperature’ component connected to the surface port by a ‘ConvectionConn’ connection.



**Main**

---

<b>Temperature</b>	Convection temperature to apply to the surface port.
<b>Convective Heat Transfer Coefficient</b>	Convective heat Transfer coefficient between the surface port and the temperature defined in the previous attribute.
<b>Surface Area</b>	Surface area over which the boundary condition should be applied. Typically this is left at “def” which will use the geometric area of the selected surface defined in the thermal mass or finite element. But if a value is entered directly, that value will override the area of the mass or finite element.



## **CHAPTER 5: GEM3D Examples**

A set of example models is provided with each installation to help illustrate the use of different templates and features. They are located in the %GTIHOME%\ <version>\examples\z\_CAD\_and\_3D\_Pre-Processing\_Tools directory of the GT-SUITE installation. They can also be opened from the File-Open Examples menu in GT-ISE.

A summary of each example has been included in each model to describe the purpose of the model, how features and templates are being used, and what modeling or design function is being achieved.



# INDEX

## 3

3 Points Cutting Plane, 18  
3 Points Datum Plane, 12, 14, 16

## A

ACIS, 86, 96, 307  
Actuator, 24, 125  
Add Arc, 66  
Add Connection, 11  
Add Point/Line, 65  
Aftertreatment, 10  
Angle, 21  
Area, 66  
Arrange Windows, 9  
Assembly Connection, 11, 140  
Assembly Rotation, 25, 28  
Axis  
  Hide, 7  
  Show, 7

## B

Baffle, 24  
  Leakage, 144  
Baffle Leakage, 24  
Bend Pipe, 9  
BiRadial, 13  
Black diamond, 19  
Boundary Manager, 11, 29  
Box Selection, 6

## C

Cancel Operation, 4  
Cap, 23  
Case Setup, 6, 30  
Catalyst, 10, 156  
CatalystBrick, 156  
CATIA, 86  
Center canvas, 66, 229, 260  
Center of Gravity, 21, 32  
center of mass, 32  
Center polygon, 67  
CG, 32  
Child Datum Plane, 12, 13, 15  
Circle, 13  
Clear All, 66



Clip, 18  
Close All, 3  
Close Model, 2, 3  
Compare Files, 23  
Component Flow Connection, 11  
Component Menu, 23  
Component Rotation, 25, 33  
Connection, 29  
  Assembly, 140  
Control Line, 22  
Control Point  
  Component, 22  
  Global, 22  
Control Points, 21  
ControlLine, 127  
ControlPoint, 128  
CONVERGE Lits, 22  
Conversion  
  Remove data, 19  
Convert Connection, 26  
Convert Menu, 16  
Convert shape, 19  
Convert Shape, 34  
Convert Shape Wizard  
  3D Tank, 59  
  Flowsplit Conversion, 43  
  Flowsplit Conversion 3 Ports, 40  
  Flowsplit Conversion 4 Ports, 42  
  Mechanical Finite Element Conversion, 62  
  Mechanical Rigid Body Conversion, 63  
  Miter Bend Conversion, 44  
  Multiple Pipes (Parallel), 57  
  Perforate Feature Conversion, 38  
  Pipe Conversion, 36  
  Pipe-T-Pipe Conversion, 54  
  Shell Conversion, 45  
  system of Flow Splits, 53  
  System of Pipes and Flow Splits, 50  
  Thermal Finite Element Conversion, 61  
  Thermal Mass Conversion, 60  
Convert to Mesh Shape, 19  
COOL3D, 22  
Copy, 5  
  Advanced Copy, 4  
Cross Section Editor, 65  
Cross section ruler, 67, 230, 260  
Cross section unit, 67, 230, 260  
Cross Sections  
  BiRadial, 13

Bi-Radial, 129  
 Circle, 13, 130  
 Custom, 13, 131  
 Ellipse, 13, 132  
 Imported, 135  
 Rectangle, 13, 133  
 Rounded Rectangle, 13, 134  
 Crossover pipe, 10  
 CustomDimension, 136  
 Cut, 5  
 Cutting Plane, 68  
   3 Points, 18  
   Pipe Normal, 17  
   Restore, 18  
   Snap to Feature, 17

## D

Datum Plane  
   3 Points, 12, 14, 16  
   Pipe Normal, 12, 14, 15  
   Single Point and Vector, 12, 14, 16  
   Snap to Feature, 12, 14, 16  
 DatumPlane, 137  
 De-convert component, 19  
 Delete, 17, 66  
 Delete Unused  
   Objects, 23  
   Objects and Template, 23  
   Templates, 23  
 Diesel Particulate Filter, 244  
 Dimension Menu, 20  
 Dimensioning, 20  
 Disconnect Components, 11  
 Discretization, 6  
 Distance, 21  
 DPF, 10, 244  
 Draft Mode, 8  
 Drill Mode, 6

## E

Edit, 5  
 Edit Flowsplit Ports, 24  
 Ellipse, 13  
 Engine Block, 327, 329, 331, 333  
 Examples, 373  
 Excel Spreadsheets, 23  
 Exit, 3  
 Export  
   ACIS, 3, 84

  gtm, 6, 70  
   gtsub, 6, 70  
   Image, 3, 69  
   Model, 70  
   STL, 3, 83  
 Export GT Model, 6  
 Extruded Connection, 11

## F

FESharedNodes, 139  
 File Menu, 2  
 Filter shapes, 19, 85  
 Find Template, 5  
 Find Value, 5  
 Fit to screen, 66, 229, 260  
 Flip Horizontal, 66  
 Flip Vertical, 67  
 Flow Connection, 4, 10  
 Flow direction, 25  
 Flow Menu, 9  
 Flowsplit, 10

## G

Gallery, 54  
 Gasoline Particulate Filter, 244  
 GEMAssyConn, 140  
 GEMBaffle, 141  
 GEMBaffleLeak, 144  
 GEMBPipe, 147  
 GEMCap, 155  
 GEMCatalystBrick, 156  
 GEMChamber, 165  
 GEMConductanceConn, 169  
 GEMConnection, 166  
 GEMConvectionConn, 167  
 GEMCrankshaft, 171  
 GEMFESharedNodes, 174  
 GEMFESurfaceMesh, 175  
 GEMFEVolumeMesh, 176  
 GEMFlowDirection, 178  
 GEMFlowsplit, 186  
 GEMFlowSystem, 179  
 GEMFsplitGeneral, 193  
 GEMInertia3D, 200  
 GEMMechSolid3D, 203  
 GEMMeshShape, 204  
 GEMMeshShell, 205  
 GEMMiterBend, 213  
 GEMMultiplePipe, 217



GEMOrifice, 224  
 GEMOrificeBaffle, 227  
 GEMOrificePipe, 231  
 GEMOrificePipeBend, 234  
 GEMOverPipe, 237  
 GEMParticulateFilter, 244  
 GEMPerfAll, 253  
 GEMPerfCS, 257  
 GEMPerfPoint, 261  
 GEMPerfRef, 265  
 GEMPerfRefBend, 269  
 GEMPipeXYZPoints, 273  
 GEMShell, 282  
 GEMSleeve, 290  
 GEMSleeveBend, 296  
 GEMSolidFlowVol, 302  
 GEMSolidShape, 307  
 GEMSolidShell, 308  
 GEMSPipe, 315  
 GEMSubAssExternalConn, 323  
 GEMTank3D, 324  
 GEMTank3DPort, 326  
 GEMThermalFE, 327  
 GEMThermalFEPort, 329  
 GEMThermalMass, 331  
 GEMThermalMassPort, 333  
 GEMTsplit, 335  
 GEMVanePump, 342  
 GEMWoolAll, 346  
 GEMWoolChamber, 347  
 GEMWoolShell, 348  
 GEMXsplit, 349  
 GEMYsplit, 356  
 Gerotor Pump, 10  
 GerotorPump, 364  
 Global Axis, 7  
 Global Datum Plane, 12, 13, 15  
 GPF, 10, 244  
 GT-ISE, 22  
 GT-POST, 22

## H

Home Menu, 3  
 Home View, 6  
 Horizontal Symmetry, 67

## I

IGES, 86, 94, 204  
 Implicit Object Links



Break All, 23  
 Refresh, 23  
 Import 3D, 3, 86  
 Import Excel Objects, 23  
 Import Shell, 86, 88, 89, 91, 94, 96  
 Internal, 20, 98

## J

JT, 86

## L

Leakage, 144  
 Length, 20, 99  
 LocalOrigin, 366  
 Locate Connection on Tree, 25

## M

Mark  
     Face, 17  
     Mark, 16  
     Redo, 17  
     Surface, 17  
     Triangle, 17  
     Undo, 17  
 Material  
     Volume, 21  
 Material Volume, 100  
 Measure  
     Diameter, 20  
     Distance, 20  
     Length, 20, 99  
     Material Volume, 21, 100  
     Surface Area, 20, 110  
     Volume, 20, 98  
 Measure Length, 99  
 Mechanical Menu, 15  
 Merge meshes, 19  
 Mesh  
     filter, 85  
     Filter, 19  
     Select all, 19  
 Mesh Flowsplit Editor, 101  
 Mirror Operation, 4  
 Model Sectioning, 6  
 Model View Layout, 9, 102  
 Move, 25  
 Move Component, 11  
 Move Group, 65

Move Point-to-Point, 25  
Multiple Pipe, 10

## **N**

New Model, 3  
Normal Mode, 8  
Notes, 6  
NX, 86

## **O**

Open Model, 2, 3  
Options, 2  
Orientation, 6  
Orifice, 23  
Overlapping Pipe, 10, 237

## **P**

Paint  
    Face, 17  
    Paint, 16  
    Redo, 17  
    Surface, 17  
    Triangle, 17  
    Undo, 17  
Parallel Baffle, 24  
Parallel Baffle to Baffle, 103  
Parallel Datum Plane, 12, 14, 16, 104  
Parameters, 6, 23, 30  
Parasolid, 86, 307  
Particulate Filter, 10, 244  
Paste, 5  
    Advanced Paste, 4  
Patch mesh ports, 19  
Perforate All, 24  
Perforate Point, 24  
Perforate Section, 24  
Perforates, 38  
Perpendicular Baffle, 24  
Perpendicular Datum Plane, 13, 14, 16  
Perspective View, 8  
Pipe Inside Pipe, 10, 237  
Pipe Normal Cutting Plane, 17, 108  
Pipe Normal Datum Plane, 12, 14, 15  
Pipe System, 9, 273  
Pipe XYZ, 9, 273  
Polygon Vertices, 66, 105  
Porosity, 20  
Print, 106

Print Preview, 107  
Pro/E, 86  
Pumps, 10

## **R**

Rail, 54  
Rectangle, 13  
Redo, 4, 65, 229, 260  
Reference Objects, 23  
Refresh, 3  
Reload, 3  
Remove deconversion data, 19  
Render Mode, 8  
Restore cutting plane, 18  
Rotate, 6, 66, 230, 260  
Rotation, 4, 25  
    Component, 25  
Rounded Rectangle, 13

## **S**

Save As, 2, 3  
Save Model, 2, 3  
Scale Component, 25  
Select all shapes, 19  
Selection  
    Body, 5  
    Box, 6  
    Chamber, 5  
    Control Elements, 5  
    General, 5  
    Line, 5  
    Quick Drill, 5  
    Vertex, 5  
Sensor, 24, 367, 372  
Separate by Curves, 18  
Set Anchor Point, 67  
Set Rotation Point, 8, 109  
Shell, 9  
Show Drilled Holes, 8  
Single Point and Vector Datum Plane, 12, 14, 16  
Sleeve, 23  
Snap to Feature Cutting Plane, 17  
Snap to Feature Datum Plane, 12, 14, 16  
Snap to Grid, 229, 260  
Solid  
    filter, 85  
    Filter, 19  
    Select all, 19  
Solid Model, 307



Spaceclaim, 86  
SPACECLAIM, 22  
STEP, 86, 307  
STL, 86, 88, 89, 91, 204  
Stop Operation, 4  
Straight Pipe, 9  
Subassembly Connection, 29  
Surface Area, 20, 110

## **T**

T Split, 9  
Table Edit View, 8  
Tank 3D Port, 24  
Template Library, 5  
Thermal, 327, 329, 331, 333  
Thermal Menu, 13  
Tile, 9  
Toggle Grid, 65, 229, 260  
Toggle Snap to Grid, 229, 260  
Toolboxes, 9  
Tools Menu, 22  
Tools>Options, 111, 112, 113, 114, 115, 116,  
118  
Translation, 4, 25, 121

## **U**

Undo, 4, 65, 229, 260  
Unselect All, 6

## **V**

Validate Drawing, 66  
Vane Pump, 10  
Vertical Symmetry, 67  
View Menu, 6  
View Model Sectioning, 122  
Volume, 20, 21, 98, 100  
VTDESIGN, 22

## **W**

Wool, 24

## **X**

X Split, 10

## **Y**

Y Split, 9

## **Z**

Zoom, 7  
Zoom Default, 66, 229, 260  
Zoom In, 66, 229, 260  
Zoom Out, 66, 229, 260

