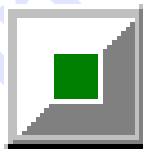


GT-SUITE

COOL3D 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
- Web Address: www.gtisoft.com
- Address: 601 Oakmont Lane, Suite 220
Westmont, IL 60559
USA

Telephone Support Hours

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

Table of Contents

CHAPTER 1: Introduction	1
CHAPTER 2: COOL3D Menu Items.....	2
File Menu.....	3
Home Menu	3
View Menu	6
Build Menu	8
Convert Menu	9
Tools Menu.....	13
Component Menu	14
CHAPTER 3: COOL3D Dialogs.....	16
Assembly Rotation – Rotates an entire assembly of connected components	17
Case Setup – Organizes cases and defines parameters	18
Convert Shape to Component – Converts an Imported Shape into a COOL3D component	20
Convert Shape Wizard – Flowspace Conversion.....	21
Convert Shape Wizard – Blockage Conversion.....	23
Cross Section Editor – 2D editor used to create and modify custom cross sections	25
Cutting Plane Control Window – Controls the direction and orientation of the cutting plane.....	28
Export gtm – Export model file (*.gtm) for use in GT-ISE.....	29
Export Image – Export graphical view(s) to an image file.....	34
Export STL – Export model to an STL file	35
Export ACIS – Export model to an ACIS file	36
File>Options>General – Contains general options and preferences specific to the application.....	37
File>Options>Favorites – Contains options regarding favorite folders and applications	39
File>Options>User Object Libraries – Contains the location of the user object libraries (.gto)	40
File>Options>Save – Contains options and preferences regarding saving models	41
File>Options>Default Units – Contains the default units preferences	42
File>Options>Default Colors – Contains the default colors preferences	43
File>Options>Conversion – Contains the mesh conversion preferences	44
File>Options>Discretization – Contains the discretization preferences.....	45
Filter Meshes – Filter mesh shapes.....	47
Import 3D – Import 3D file	48
Import ACIS – Import an entire ACIS file	49
Import IGES – Import an entire IGES file.....	51
Import Parasolid – Import an entire Parasolid file.....	53
Import STL – Import STL file	55
Import STL as Cross Sections – Import a shell from an STL file using cross sections.....	56
Import STL as Surface – Import an entire STL file from its surfaces	59
Local Cutting Plane Information Window – Provides the orientation of the local cutting plane.....	61
Model View Layout – Controls the layout of the model in the display window	62
Polygon Vertices – Allows creation and editing of custom cross section shape by coordinate points..	63
Set Rotation Point – Sets the anchor point for graphical rotations	64
Translation – Translates a component	65
View Model Sectioning – Graphically sections the model so that interior components can be viewed	66

Table of Contents

CHAPTER 4: COOL3D Components.....	68
COOLBlockage	69
COOLDuct.....	71
COOLFan	73
COOLFlowResistance	78
COOLFlowSpace.....	80
COOLFlowSpaceSimple	83
COOLHeatAddition.....	85
COOLHeatExchanger	89
COOLSolidBlockage	97
COOLSolidFlowSpace	98
DiscretizationPlane	101
GEMMeshShape – General Mesh Shape.....	102
GEMSolidShape - General Solid Shape	103
CHAPTER 5: COOL3D Features	104
ActuatorConn3D - Actuator Connection	105
FlowOpening	107
HoleDuct.....	110
HoleVolume.....	111
HoleVolumeObject	112
HoleVolumeZero	113
SensorConn3D - Sensor Connection	114
CHAPTER 6: COOL3D Cross Sections	119
CSBiRadial – Bi-Radial Cross Section.....	120
CSCircle – Circle Cross Section.....	121
CSCustom – User Defined Cross Section.....	122
CSEllipse – Elliptical Cross Section.....	123
CSRect – Rectangular Cross Section.....	124
CSRoundRect – Rounded Rectangular Cross Section.....	125
CHAPTER 7: COOL3D Reference Templates	126
COOLEndEnv – Pressure Boundary	127
COOLEndEnvRam – End Environment with Ram Velocity	132
COOLEndFlowInlet – Imposed Flow Rate	137
EffAreaRestriction.....	140
XYTableSimple	141
Index.....	142

CHAPTER 1: Introduction

COOL3D is a tool that can be used to build 3D models of underhood thermal management 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. COOL3D can be used to build any flow system that contains only blockages, fans, heat additions, and heat exchangers.

This manual describes the operations and components available in COOL3D. It is assumed that the user to the user's

is already familiar with the GT-SUITE software package including GT-ISE and GT-POST. COOL3D 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 COOL3D and can be a useful reference, but it is strongly recommended to attend a training class. Since COOL3D is a new tool in GT-SUITE, specific training classes will be available on COOL3D. 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-gt/contact-by-territory/> for contact information for our representatives in Japan, Korea, China, India, and Brazil.



CHAPTER 2: COOL3D Menu Items

The COOL3D menu items section contains descriptions for each menu item available in COOL3D. This also includes all operations and commands that may be available in COOL3D on right-click menus and shortcut keys. These descriptions can be found in the context help while using COOL3D 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 COOL3D 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 COOL3D model file will have a *.ghx* 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: Send the selected document in an email, or contact GT Support.

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 COOL3D 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](#) dialog.



Exit: Exits the COOL3D application. [Keyboard hotkey = alt+F4]

Home Menu

The home menu includes the most commonly needed operations for managing COOL3D models.



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





GT Object Library: Create a new object library file.



Open: Allows the user to open an existing model file. A COOL3D model file will have a *.ghx* extension. [Keyboard hotkey = ctrl+O]



Open and export: Allows the user to open an existing model file and export it as a Parasolid file (.x_b) simultaneously.



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]



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 COOL3D 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 selection mode enables components to be selected to access other operations (i.e. Component menu).



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 GTM: Discretizes the active model and exports a model file (.gtsub 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.



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.

Build Menu

The build menu contains all the operations used to build components.



Flow Space: Creates a flow space where the geometry is defined by the user. This specifies the air domain of the model, and must be created before the model can be exported to GT-ISE. For more information, see the help for the '[COOLFlowSpace](#)' template.



Flow Space Simple: Creates a simple flow space where the geometry is automatically defined by the components in the model. This specifies the air domain of the model, and must be created before the model can be exported to GT-ISE. For more information, see the help for the '[COOLFlowSpaceSimple](#)' template.



Heat Exchanger: Creates a heat exchanger. For more information, see the help for the '[COOLHeatExchanger](#)' template.



Heat Addition: Creates a heat addition. For more information, see the help for the '[COOLHeatAddition](#)' template.





Blockage: Creates a blockage. For more information, see the help for the '[COOLBlockage](#)' template.



Fan: Creates a fan. For more information, see the help for the '[COOLFan](#)' template.



Duct: Creates a duct. For more information, see the help for the '[COOLDuct](#)' template.



Flow Resistance: Creates a flow resistance (pressure drop) plane. For more information, see the help for the '[COOLFlowResistance](#)' template.



Circle: Creates a circular cross section. For more information, see the help for the '[CSCircle](#)' template.



Rectangle: Creates a rectangular cross section. For more information, see the help for the '[CSRect](#)' template.



Round Rectangle: Creates a rectangular cross section with rounded corners. For more information, see the help for the '[CSRoundRect](#)' template.



Ellipse: Creates an elliptical cross section. For more information, see the help for the '[CSEllipse](#)' template.



BiRadial: Creates an irregular elliptical cross section. For more information, see the help for the '[CSBiRadial](#)' template.



Custom: Creates a custom cross section defined by selected points. For more information, see the help for the '[CSCustom](#)' 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.



Local 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 Local Cutting 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 [Tools>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.

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.



GEM3D: Starts the GEM3D 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 contains operations that can be performed on components only. This menu will appear only when a component is selected. Most items found in this menu can be accessed by right-clicking on a component.



Flow Opening... : Creates a flow opening (i.e. flow boundary condition). For more information, see the help for the ['FlowOpening'](#) template.



Duct Holes: Creates a hole or flap in a ['COOLDuct'](#) component. For more information, see the help for the ['HoleDuct'](#) template.



Blockage Holes: Creates a hole or flap in a ['COOLBlockage'](#) component.



Hole With Volume... : Creates a hole in a blockage that includes the fluid volume. For more information, see the help for the ['HoleVolume'](#) template.



Hole With Flaps or No Volume... : Creates a hole that will have no fluid volume, or a flap. For more information, see the help for the ['HoleVolumeZero'](#) template.



Hole With COOL* Object... : Creates a hole in a blockage that is taken up by some other COOL* object (heat exchanger, heat addition, or fan). For more information, see the help for the ['HoleVolumeObject'](#) template.



Discretization Plane... : Creates a discretization plane that belongs to the flow space for additional slices along the normal direction. For more information, see the help for the ['DiscretizationPlane'](#) template.



Sensor : Creates a sensor to sense flow conditions (i.e. pressure, temperature, etc). 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.



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



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





CHAPTER 3: COOL3D Dialogs

The COOL3D dialogs section contains information on operations in COOL3D 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 COOL3D 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.
- 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 COOL3D 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 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

- | | |
|--------------|--|
| Angle | 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.





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 COOL3D 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(θ)

min, max(a,b)

acos, asin, atan()

floor, ceil()

pi()

exp(a)

sqr, sqrt()

ln()



	1	2
a		
x		

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.





Convert Shape to Component – Converts an Imported Shape into a COOL3D component

This operation is used to convert a solid shape ('[GEMSolidShape](#)') or mesh shape ('[GEMMeshShape](#)') into a COOL3D3D component. This must be done for all mesh shapes that need to be discretized because a mesh shape will not be discretized during an [Export .gtm](#) operation.

Convert Mesh

Convert to

Specifies the type of flow component that the imported shape will be converted into. Available components include:

- **Flow Space:** Specifies the mesh shape will be converted into a flow space ('[COOLFlowSpace](#)').
- **Blockage:** Specifies the mesh shape will be converted into a blockage ('[COOLBlockage](#)').





Convert Shape Wizard – Flowspace Conversion

COOLFlowSpace

The conversion to 'COOLFlowSpace'. All detected ports are automatically neglected when converting to a Flow Space. A closed volume will be the result of this conversion

Cross Section Type

- **Create cross sections** creates a series of cross sections to approximate the imported shape. These cross sections can be customized, see the sections below for more details
- **Use Imported Shape** will use the existing geometry as the Flow Space shape. This option is only available when the original geometry is imported as a Solid Shape.

Cross Section Definition:

- **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 flow space easier to work with.

Specify Cross Sections

These options are only available with "Create cross sections":

- **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 flow space. 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 flow space ends. The number of cross sections requested must be between 4 and 200. The cross sections will be taken along the X axis. The cross sections will be evenly





spaced along the X 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 flow space.

The cross sections at the ends of the converted flow space 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.

- **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.





Convert Shape Wizard – Blockage Conversion

COOLBlockage

The conversion to 'COOLFlowSpace'. All detected ports are automatically neglected when converting to a Blockage. A closed volume will be the result of this conversion

Cross Section Type

- **Create cross sections** creates a series of cross sections to approximate the imported shape. These cross sections can be customized, see the sections below for more details
- **Use Imported Shape** will use the existing geometry as the Flow Space shape. This option is only available when the original geometry is imported as a Solid Shape.

Cross Section Definition:

- **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 flow space easier to work with.

Specify Cross Sections

These options are only available with "Create cross sections":

- **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 blockage. 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 flow space ends. The number of cross sections requested must be between 4 and 200. The cross sections will be taken along the X axis. The cross sections will be evenly





spaced along the X 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 blockage.

The cross sections at the ends of the converted blockage 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.

- **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.





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 Snap to 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.



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.



Flip Vertical: Flips the cross section vertically.



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.



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.



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, and ft.



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.

Cutting plane control

Location	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).
Orientation	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).
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.
Rotate X	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 Rotate plane option below.
Rotate Y	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 Rotate plane option below.
Rotate Z	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 Rotate plane option below.
Translate plane	Drop down option that determines how far the cutting 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 cutting plane rotates with each move of the Rotate sliders above. Allowed values range from 1 to 10.

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 gtm – Export model file (*.gtm) for use in GT-ISE

This operation is used to discretize the COOL3D model and export a *.gtm model file for use with GT-SUITE.

Discretization

Model license type	The file type, GT-SUITE or GT-SUTIE-MP , for the discretized .gtm model.
Output file name	Specifies the file name and location of the *.gtm file to be exported. Also allows a browse feature to choose the location and file name of the exported *.gtm file. By default the COOL3D model name will be used as the output file name.
Append to file name (multiple cases)	<p>Select an option below to adjust the filename of the discretized model when working with multiple cases.</p> <ul style="list-style-type: none"> • 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.

Advanced Options

Use improved fan discretization rule	When checked, fans in COOL3D will be discretized to conserve the volume and flow area swept by the fan, along with changing the flow multiplier that is calculated for each fan element. The flow multiplier that is calculated is directly proportional to the flow area of the fan element.
---	---

Preview Options

Preview options	<p>Specifies 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. Available preview options include:</p> <p>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.</p> <p>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.</p> <p>Show cube with mesh line displays both the cubes and the mesh lines.</p>
------------------------	---





	Show mesh line displays lines that represent the edges of each flowsplit.
	Show mesh face displays planes (faces) the 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 objects will be shown in the graphical window with the discretization preview. When not checked, the model objects will not be shown.
Show air matrices	Toggle to display the discretized cubes for the flow space during a discretization preview. When checked, the flow space cubes will be shown in the graphical window with the discretization preview. When not checked, the cubes for the flow space will not be shown.
Show HX/HA matrices	Toggle to display the discretized cubes of heat exchangers and heat additions during the discretization preview. When checked, the heat exchanger and heat addition cubes will be shown in the graphical window with the discretization preview. When not checked, the cubes for these objects will not be shown.

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 the COOL3D model.

NOTE: Objects that are not checked in the model tree will not be included in the discretized model. In order to include object in the discretization they must be checked.

Once the discretization parameters have been filled in, the flow space is divided into slices in the global X-direction as air grids. Each air grid will begin and end in between object layers. Each slice is further subdivided into rectangular sections based on the number of horizontal and vertical discretization cells. This results in the flow space being divided up into many boxes that approximate the flow path area. Once the number of rectangular sections is determined, the total volume in that slice (minus the volume of any internal objects) is divided equally among the sections. An exception to this is when a rectangular box overlaps an object, in which case the volume and area of that box is modified to represent the flow path only (see [Flowsplit acceptance ratio](#)). The resulting 3D information is placed on the GT-SUITE project map as 'MatrixFlowSplit' parts in a manner that visually represents their location in 3D space. Flow elements within the flow space will be created as described in the following paragraphs, and will be placed on the project map in the order they are found along the global X-axis.

It may be necessary to manually choose a discretization point along the flow direction (global X-direction) for the flow space. This would provide the same discontinuity for the discretization that objects do, however, without creating a physical division and disturbing the flow path. This is useful when more detail is required in the flow space. To handle this, the discretization routine will use discretization planes





('DiscretizationPlane' template) as a reference point when discretizing and start calculating a new air grid in the flow direction perpendicular to the discretization plane.

Rather than discretize a 'COOLBlockage' object, the air grid in which it lies in is discretized to represent an obstruction to flow. Any solid obstacle will prevent the connection of 'MatrixFlowSplit' objects between adjoining air grids upstream and downstream from the blockage, as well as preventing any lateral connections within the air grid to represent a solid object. Any openings in the blockage to represent holes will allow the connection of subvolumes upstream, downstream, and laterally to model an opening in the object. Additionally, any holes may have an attached 'FlowPDDrop' to model the pressure loss through an opening. This pressure loss will be scaled appropriately with a multiplier that is equivalent to the available area of the cube divided by the total area of the opening.

The 'COOLFan' object is discretized according to the air grid where a flow path exists through the shroud. If the flow path is through the fan, this results in a 'MatrixFan' object that contains the 3D information of the parent object. A flow rate multiplier is imposed on each object to discretize the total flow into the smaller elements. A sub-volume that includes part of the fan and part of the shroud, or hub, will have a smaller flow rate multiplier for its subsequent discretized part, whereas a sub-volume that includes only the fan will not. Additionally, an 'XYTableSimple' reference object can be used to normalize the radial distribution of the air flow through the fan. This will result in another flow rate multiplier to be taken into account. The sum of the flow rate multipliers will be 1. If the flow path is through a flap or hole, this will result in connections representing the opening. If the opening is to be a hole, the connection to the subvolumes on either side of the shroud will be a default orifice connection. If the opening is to be a flap, the connection to the subvolumes on either side of the shroud will be a 'ValveCheckSimpleConn'. Modifications to the [Flowsplit acceptance ratio](#) may change the number of fan elements and connections created.

The 'COOLHeatAddition' object has its own discretization that the user defines separately from the discretization of the flow space. When discretized, the volume and heat input is scaled proportionally to the number of elements created in the 'MatrixHA' part to represent the full object. The sum of the total pipe area and the total heat input of every discretized object will equal the parent object. Additionally, a multiplier is used to scale the external pressure loss object for each sub-volume to model the pressure loss through the discretized heat addition object.

The 'COOLHeatExchanger' object has its own discretization that the user defines separately from the discretization of the flow space. Once discretized, a 'MatrixHx' part is created that contains the 3D geometry to model the heat exchanger characteristics for both the internal and external sides. Additionally, a multiplier is used to scale the external pressure loss object for each sub-volume to model the pressure loss through the discretized heat exchanger object. A single pressure loss object is used to model the internal pressure drop of the heat exchanger.

Any open ports in the model file that require a default connection will be connected to a subassembly connection once it is 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 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 attached to a heat exchanger to define the coolant flow, or a rigid connection at the end of shaft for a fan to define the speed boundary rotation.

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 .ghx 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: **cool3dc [OPTIONS] filename**
 Specific format: **cool3dc -VM.m.s -o:filename.gtm -p:parametername=value filename.ghx**
 Example: **cool3dc -V7.3.0 -o:output.gtm -p:diameter=50 model.ghx**

Available [OPTIONS]:

- help** Shows script help. This option is used only by itself, without other options or a filename. For example, to use the help option one might type **cool3dc -h**.
- 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, 7.3.0.
- o:filename** Specifies the filename to be used for the output file created by COOL3D. 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 .ghx file (just with a .gtm extension).
- 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 COOL3D 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).
- m** Specifies that a 3D model file (.ghx) will be created for each configuration. The model file name will be taken from a configuration file (if used) or the input COOL3D file. This option must be used carefully as it may overwrite the existing .ghx file that was used as an input. This option is ONLY intended to be used with the configuration file option above.
- 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
- 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 COOL3D model. This option is useful when trying to create multiple discretized models from a single COOL3D to test different components, such as heat exchangers. The general format of





the configuration file is below.

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 GT 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 applied to attributes that have units assigned to them (ex. location).

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.gtm = case1.gtm	{ output filename with .gtm extension of discretized model }
Flowspace.FLOWDX = 100 mm	{ global discretization of the X-axis }
Flowspace.FLOWDY = 100 mm	{ global discretization of the Y-axis }
Flowspace.FLOWDZ = 100 mm	{ global discretization of the Z-axis }
Flowspace.FSRATIO = 0.5	{ acceptance ratio of the global discretization }
Radiator.LOCX = 250 mm	{ changing the X position of the Radiator object }
...	...
Fan.state = off	{ remove the Fan object from the discretization }
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 }
Radiator.NHXDX1 = 25 mm	{ axial discretization for the Radiator object }
Radiator.INITSTATE1 = Coolant_Initial	{ name of a new internal initial condition for the Radiator }
CAC.state = off	{ remove the CAC object from the discretization }
...	...
OilCooler.state = on	{ add the OilCooler object to the discretization }

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 Image – Export graphical view(s) to an image file

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

Export active view as image	Exports the currently active view to an image file.
Export all views in separate images	Exports all views in the graphical display to separate image files, 1 for each view.
Export all views in a single image	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.
White Background	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.
Keep Ratio	When checked, the ratio of the Width and Height 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).
Width	Width, in pixels, of the exported image.
Height	Height, in pixels, of the exported image.
Output file	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 COOL3D 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.



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.

Output file name/location	Specifies the file name and location of the STL file to be exported. The Output To button allows selection of the location and file name using a file browser dialog window. When the Output separate STL files option is not checked, this field specifies the output file name and its location. When the Output separate STL files option is checked, this field only specifies the output files location.
Selected components only	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.
Output separate STL files	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.
Resolution	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 cannot be exported to ACIS format.

Output file name/location	Specifies the file name and location of the ACIS file to be exported. The Output To button allows selection of the location and file name using a file browser dialog window. When the Output separate files option is not checked, this field specifies the output file name and its location. When the Output separate files option is checked, this field only specifies the output files location.
Selected components only	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.
Output separate files	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.



File>Options>General – Contains general options and preferences specific to the application

This menu contains options that can be used to control how the COOL3D application operates.

General

Auto display toolboxes	When this option is checked, COOL3D will automatically open and position all toolboxes that were open the last time the application was closed.
Icon only on button	When this option is checked, COOL3D will only display the icons of operations in the toolboxes. When un-checked, COOL3D will display the icon and the name of the operation.
Do not show Start Up Window	If checked, the start up window will not be displayed when the application is launched.
Issue warning if objects are off when file is opened	If selected, a warning message will be issued upon opening a file that was saved with objects in the "off" position.
Display Marker for X,Y,Z location	If turned on, a mouse-click will display a white marker at the location selected. This location can be seen in the lower right portion of the application window.
Do not highlight component when mouse cursor is on it	If selected, an object will not be highlighted when the mouse cursor is placed over it.
Highlight attributes that affect modeling	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).
Re-normalize directions after pressing 'Apply' or 'Ok'	When checked, all direction attributes will be automatically re-normalized after pressing 'Apply' or 'Ok'.
Definition of Component Origin	<p>The prefill value for the Component Origin Reference in 'COOLFlowResistance', 'COOLHeatAddition', and 'COOLHeatExchanger'. All locations are assumed to be looking along the +X axis, with left being +Y and top being +Z.</p> <ul style="list-style-type: none">• center: The center point of the object will be the origin.• upper right: The upper right corner of the object will be the origin.• lower right: The lower right corner of the object will be the origin.• upper left: The upper left corner of the object will be the origin.• lower left: The lower left corner of the object will be the origin.



- **top center:** The top center point of the object will be the origin.
- **left center:** The left center point of the object will be the origin.
- **bottom center:** The bottom center point of the object will be the origin.
- **right center:** The right center point of the object will be the origin.

Number of significant figures displayed

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.

Default Transparency Percentage [0-90]

Indicates the default transparency level used when an individual component has a **Transparency Percent** of "def". 0 indicates opaque (solid) and 90 indicates almost completely transparent.

Move to next cell on enter:

When pressing 'Enter' to validate a cell entry, the focus of the cell will follow the selection made here.

Refresh implicit objects when opening .gtm, .gem, and .ghx files

Refresh all implicit links automatically when a file is opened.



File>Options>Favorites – Contains options regarding favorite folders and applications

This tab contains options regarding favorite folders and applications.

Favorites

Default Template Library

Default template library to be used by COOL3D.

Default Editor

Default text editor used by the application. When this is left blank, the application's default text editor is Notepad on Windows.

Favorite Folder



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>User Object Libraries – Contains the location of the user object libraries (.gto)

This tab contains options regarding the user object libraries.

Favorites

User Specified Libraries The location of the user libraries (.gto) which will be preloaded into the application when it is launched.



File>Options>Save – Contains options and preferences regarding saving models

This tab contains options that can be used to control how COOL3D saves models.

Save

AutoSave every ____ minutes

When this option is checked, each open *.ghx 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 *.ghx 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, COOL3D 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 COOL3D application.

Default Units

Length	Default length unit used by the application.
Area	Default area unit used by the application.
Volume	Default volume unit used by the application.
Angle	Default angle unit used by the application.
Inverse area	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 COOL3D application.

Default Colors

Default Component/Feature Display Color	The default color selected here will be the default color selected in the Visual folder when a template is first created.
Background Color	The color selected here will be the background color for the canvas.
Highlight Color	The color selected here will be the color displayed when the mouse pointer is over an object. This color will not be visible if the Do not highlight component when mouse cursor is on it is selected in the General folder of File>Options .
Select Color	The color selected here will be the color displayed when an object is selected.
Wheel Color	The color selected here will be the color of the display widget.



File>Options>Conversion – Contains the mesh conversion preferences

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

Conversion

Allow Mesh Deconversion

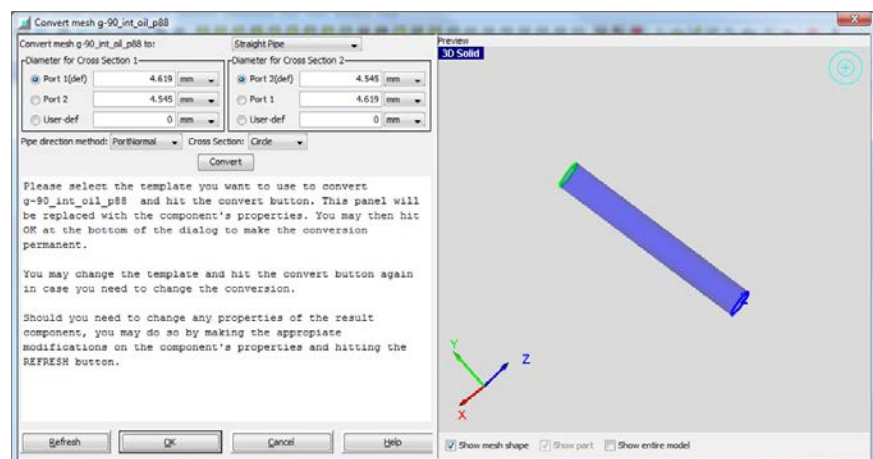
When checked, this option will allow a COOL3D component converted from a mesh shape to be de-converted back into its original mesh shape. The advantage to this option is the ability to go directly back to the mesh shape if the conversion options need to be changed or the mesh shape modified. The disadvantage 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 a flow space or a blockage. This option will apply to all attributes including location, distance to next cross section, etc. By default, three digits are displayed after the decimal, as shown in the example conversion dialog below.



Mesh Preview Transparency Percent

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



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 COOL3D 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, COOL3D 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.

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 flow splits are needed. Then, it will display all the flow splits 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 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 cube** displays each flow split 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 flow split 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 flow split.
- **Show mesh face** displays planes (faces) the represent the division between flow splits.
- **Show mesh face and line** displays both the mesh faces and lines.

Cube transparency level (%)

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 [Export .gtm](#) operation.

Object Names - Prefix

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 COOL3D 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.

Object Names - Suffix

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 COOL3D 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.

Part Names - Prefix

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 COOL3D 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.

Part Names - Suffix

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 COOL3D 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.



Filter Meshes – Filter mesh shapes

This operation is used to quickly filter the mesh 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 mesh shapes that are not useful in the modeling process. This includes flanges, brackets, tabs, bolts, and other features in the STL file that are not useful when trying to create a flow model of the system. This operation will pass all mesh shapes through a filter to create a list of mesh shapes that are not likely to be necessary flow components, and thus can be deleted. To fail the filter validation the mesh 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 mesh shape that does not meet the above 2 criteria.

Name Lists the name of each mesh shape that did not pass the filter validation. Clicking on a mesh shape name or names in this list will display the component as selected in the graphical window. This allows the mesh shape to be identified graphically to ensure this component needs to be filtered.

Tolerance 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 mesh shapes failing the filter validation. However, the mesh shapes that did fail have a very low probability of being actual components, thus preventing accidental deleting of mesh 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 mesh 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 mesh shape names.





Import 3D – Import 3D file

This operation is used to import a 3D exchange format file. An exchange format is a file format that is not specific to any particular software package, such that the format may be used by multiple software packages to exchange information. The allowed file types (formats) are ACIS (.sat, .sab), Parasolid (.x_t, .x_b), STL (.stl), and IGES (.iges, .igs). Other formats are supported by GT-SUITE through GT-SPACECLAIM (available in the Tools menu), see the table below.

Import 3D File

Files to Import

This field specifies the ACIS (.sat, .sab), Parasolid (.x_t, .x_b), STL (.stl), or IGES (.iges, .igs) file to be imported. Multiple files can be specified but they must all be of the same type.

The next steps in the import process will depend on the type of the imported file. ACIS files will open the [Import ACIS](#) dialog window. Parasolid files will open the [Import Parasolid](#) dialog window. STL files will open the [Import STL](#) dialog window. IGES files will open the [Import IGES](#) dialog window.

Preferred CAD Formats for Import to GT-SUITE

File Format	Instructions for import
Preferred Formats	
Parasolid, Spaceclaim, ACIS (.x_t, .x_b, .scdoc, .sat, .sab)	Files can be imported directly to GEM3D and COOL3D. GT-SpaceClaim may convert these files in the background so they can be read by GT-SUITE. Files can also be opened in GT-SpaceClaim for modification if geometry changes are desired. For Linux machines, ACIS is recommended.
Native CAD Formats	
SolidWorks	Files can be imported to GEM3D and COOL3D, and will automatically be converted to GT-SUITE format. Files can also be opened in GT-SpaceClaim if necessary.
Unigraphics	
CATIA	
Inventor	
Pro/Engineer Creo	
Data Exchange Formats	
STL	Files can be imported directly to GEM3D and COOL3D. This format is acceptable for single flow components; for compound components or conversions to finite element parts a solid model format (Parasolid, STEP, ACIS) should be used.
STEP	Files can be imported to GEM3D and COOL3D, and will automatically be converted to GT-SUITE format. Files can also be opened in GT-SpaceClaim for modification if necessary.
IGES	Files can be imported directly to GEM3D and COOL3D. Files can also be opened in GT-SpaceClaim if there are problems with the import process. It should then be saved to a solid model format and imported to GEM3D or COOL3D. Due to the possibility for missing surfaces, solid model (Parasolid, STEP, ACIS) or native CAD formats are recommended.





Import ACIS – Import an entire ACIS file

This operation is used to import the entire contents of a ACIS file into COOL3D. During the import routine, all the shapes in the file are determined and used to create the shape in COOL3D. 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 COOL3D 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 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 file should not contain wall thickness. In other words, it should represent the inside geometry. If the file does contain wall thickness, the outer wall can usually be removed with the [Separate by Curves](#) operation or the [Marking](#) operation.
- 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. If this situation exists, then the [Patching](#) operation may be able to help, as well as other clipping operations.

This operation is also available from the command line, by using the "cool3d" 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 COOL3D. 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 COOL3D. A lower resolution can be used to reduce the model size and computational load. If COOL3D 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 IGES – Import an entire IGES file

This operation is used to import the entire contents of an IGES file into COOL3D. During the import routine, all the shapes in the file are determined and used to create the shape in COOL3D. 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 COOL3D 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.
- The file should not contain wall thickness. In other words, it should represent the inside geometry. If the file does 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. If this situation exists, then the [Patching](#) operation may be able to help, as well as other clipping operations.

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 COOL3D. 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 COOL3D. A lower resolution can be used to reduce the model size and computational load. If COOL3D 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 Parasolid – Import an entire Parasolid file

This operation is used to import the entire contents of a Parasolid file into COOL3D. During the import routine, all the shapes in the file are determined and used to create the shape in COOL3D. 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 COOL3D 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 Parasolid 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 file should not contain wall thickness. In other words, it should represent the inside geometry. If the file does contain wall thickness, the outer wall can usually be removed with the [Separate by Curves](#) operation or the [Marking](#) operation.
- The Parasolid 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 [Patching](#) operation may be able to help, as well as other clipping operations.

This operation is also available from the command line, by using the "cool3d" command followed by the name of the Parasolid 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 Parasolid file into COOL3D. 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 COOL3D. A lower resolution can be used to reduce the model size and computational load. If COOL3D 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 STL – Import STL file

This operation is used to import an STL file. There are 2 different methods of importing an STL file into COOL3D.

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 COOL3D 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 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 COOL3D. This will only work to import the shell. Any interior components (like baffles and pipes) will be ignored and must be built manually in COOL3D. 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 COOL3D 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

STL File	Specifies the name of the STL file containing the shell to be imported.
Cross Sections	Specifies the number of cross sections that will be taken between the mesh ends. The number of cross sections requested must be between 4 and 200. The cross sections will be taken perpendicular to the X-Axis and will be evenly spaced along the X-Axis . The number of cross sections chosen should be sufficiently large to accurately define the shape of the shell.
Vertices	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.
Major Axis	<p>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:</p> <ul style="list-style-type: none">• def indicates the major axis will be the direction of the longest shell dimension (length, width, or height).





- **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:

- **m³**
- **cm³**
- **mm³**
- **in³**
- **ft³**
- **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 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 COOL3D. During the import routine, all the surfaces of the STL file are determined and used to create the shape in COOL3D. 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 COOL3D 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.
- 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 [Patching](#) operation may be able to help, as well as other clipping operations.

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**





Local Cutting Plane Information Window – Provides the orientation of the local cutting plane

This window provides some detailed geometrical information about the [local cutting plane](#). The local cutting plane can be controlled graphically using the mouse, or the keyboard arrow keys can be used to make small move increments.

Local Cutting Plane Info

Angle	Shows a real-time counter of the relative angle between the current orientation of the local cutting plane and the orientation of the end of the mesh shape component. If this angle is constant along the length of a mesh 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 and end of bent sections.
Effective Diameter	Shows a real-time counter of the effective diameter at the current location of the local cutting plane.
Diameter	Shows a real-time counter of the diameter at the current location of the local cutting plane.

Clip: This button will do the [Clip](#) operation. This will separate the component along the current location and orientation of the local cutting plane.





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

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





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 points. 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 of rows in the table.





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 the about the screen Z axis (Ctrl key + left mouse button + mouse drag). The anchor point is stored as an application property.

Global origin (0,0,0)

Sets the rotation anchor point to the origin. The origin is the point with global coordinates of $X = 0$, $Y = 0$, and $Z = 0$.

Global Centroid

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.

Component's Centroid

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.

Select Control Point

Allows the rotation anchor point to be set by the coordinates of a selected control point or the absolute coordinates specified by the **X**, **Y**, and **Z** 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 **X**, **Y**, and **Z** attributes below. This allows the coordinates to be checked and/or modified. The **X**, **Y**, and **Z** attributes may also be directly specified to set the rotation anchor point. The specified coordinates are in the [global coordinate system](#).

X

Absolute X coordinate in the [global coordinate system](#) 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 **Select Control Point** option is enabled above.

Y

Absolute Y coordinate in the [global coordinate system](#) 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 **Select Control Point** option is enabled above.

Z

Absolute Z coordinate in the [global coordinate system](#) 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 **Select Control Point** option is enabled above.



Translation – Translates a component

This operation is used to translate a component a specified distance in the X, Y, and/or Z directions.

- 1) To begin a translation, the component must first be selected.
- 2) Then click on the Translation operation and the Translation dialog window will open.
- 3) 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.

Translate component

X	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.
Y	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.
Z	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) Press OK or Apply to finish the translation.





View Model Sectioning – Graphically sections the model so that interior components can be viewed

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 of a flow space.

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: COOL3D Components

The COOL3D components section contains information on each of the components used in COOL3D. A description of each component 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 COOL3D by clicking the button in the upper left corner of each template. This button will have an image of the template as well as a small question mark symbol.





COOLBlockage

This object is used to model the flow resistance of an object impeding flow in a flow space.

Cross Sections

Multiple columns should be used to represent an object that changes cross section. This is useful to create objects that are not extruded along a constant path.

Cross Section Name	Name of the ' CSCircle ', ' CSCustom ', ' CSEllipse ', ' CSRect ', or ' CSRoundRect ' reference object describing the geometry of the blockage. A value of "def" will extend the boundary of the blockage to the boundary of the flow space.
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. If wanted, "def" or "ign" can be entered for the last cross section.
Relative Y Location of Center	The relative distance in the Y-direction from the center of the first cross section to current cross section. "def" will have a value of 0.
Relative Z Location of Center	The relative distance in the Z-direction from the center of the first cross section to current cross section. "def" will have a value of 0.

Location

Location X (Global)	Specifies the absolute X location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if ' CSCustom ' is used for the Cross Section Name . Figure 1 shown below has a blockage location of (0,0,0).
Location Y (Global)	Specifies the absolute Y location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if ' CSCustom ' is used for the Cross Section Name . Figure 1 shown below has a blockage location of (0,0,0).
Location Z (Global)	Specifies the absolute Z location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if ' CSCustom ' is used for the Cross Section Name . Figure 1 shown below has a blockage location of (0,0,0).
Create Holes for Intersecting Components	If turned "on", when the ' COOLBlockage ' components intersects with any ' COOLHeatExchanger ', ' COOLHeatAddition ', or ' COOLFan ' component, then a hole will be automatically created in the



'COOLBlockage' component to fit the intersecting shape.

One example allows a frame that holds a cooling module to be quickly created to prevent bypassing air flow. The holes generated for the frame will automatically adjust in size as the heat exchanger and/or fan components change shape and/or location.



Figure 1: Location of a blockage in global space at location (0,0,0).

Discretization

Parameterize Object to Case Setup	If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.
-----------------------------------	---

Visual

Transparency Percent	Indicates the transparency level used when drawing the part. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent.
----------------------	---

Display Color	Indicates the color used when drawing the object. The color choices include:
---------------	--

- | | |
|------------|---------|
| •Red | •Copper |
| •Blue | •Gold |
| •Dark Blue | •Grey |
| •Green | •Black |





COOLDuct

This template is used to model an object with thin walls that redirects flow to a new location (i.e. a duct).

Cross Sections

Multiple columns should be used to represent an object that changes cross section. This is useful to create objects that are not extruded along a constant path.

Cross Section Name	Name of the ' CSBiRadial ', ' CSCircle ', ' CSCustom ', ' CSEllipse ', ' CSRect ', or ' CSRoundRect ' reference object describing the geometry of the duct.
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. If wanted, "def" or "ign" can be entered for the last cross section.
Relative Y Location of Center	The relative distance in the Y-direction from the center of the first cross section to current cross section. "def" will have a value of 0.
Relative Z Location of Center	The relative distance in the Z-direction from the center of the first cross section to current cross section. "def" will have a value of 0.

Location

Location X (Global)	Specifies the absolute X location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if ' CSCustom ' is used for the Cross Section Name . Figure 1 shown below has a duct location of (0,0,0).
Location Y (Global)	Specifies the absolute Y location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if ' CSCustom ' is used for the Cross Section Name . Figure 1 shown below has a duct location of (0,0,0).
Location Z (Global)	Specifies the absolute Z location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if ' CSCustom ' is used for the Cross Section Name . Figure 1 shown below has a duct location of (0,0,0).



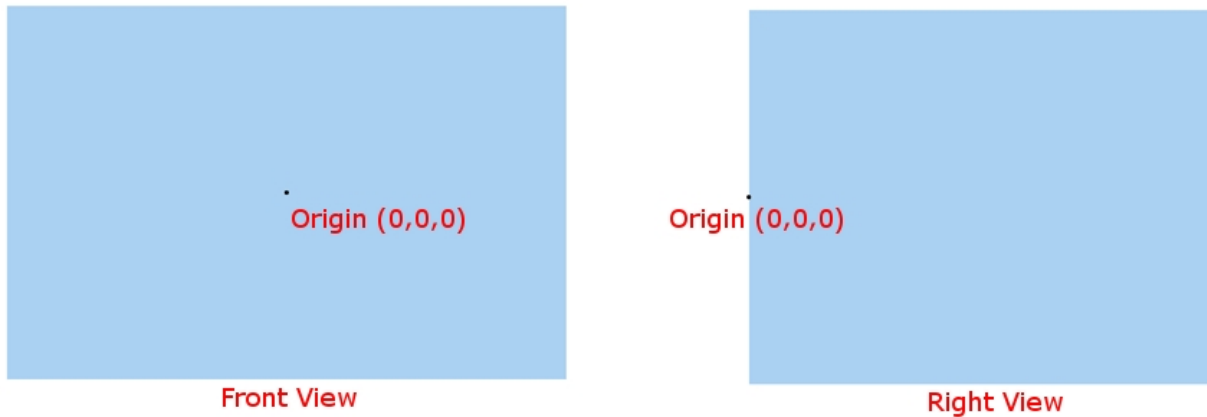


Figure 1: Location of a duct in global space at location (0,0,0).

Discretization

Parameterize Object to Case Setup

If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.

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 object. The color choices include:

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



COOLFan

This object is used to model a fan when a full map of data, fan coefficients, or equation is available to correlate the mass flow rate, fan speed, pressure rise, and efficiency ('Fan') of the fan component.

Main

"As Tested" Fan Specifications Object

Name of the 'FanSpecs' reference object that defines the fan performance and geometry as it was originally measured.

Note: GT supplies a Microsoft Excel spreadsheet that enables a 'FanSpecs' object to be directly imported into a model (or into an object library) based on the fan data entered in the spreadsheet. The spreadsheet is intended to offer an efficient method of communicating data between a fan supplier (who may not have GT-SUITE) and a user of GT-SUITE. (**Tools > GT Excel Sheets ... > Fans > FanSpecs.xlsx**)

"As Used" Fan Blade Diameter

This is the new blade diameter for the fan that will be used when scaling the fan performance data. To scale the fan performance data the original fan geometry (Blade Diameter and Hub Diameter) must have been filled out in the 'FanSpecs' reference object. Fan performance may only be scaled if the fan performance is specified using a non-dimensional approach (via 'FanMap' with the **Non-Dimensional Coefficients** option, or directly via the 'FanCoefficients' or 'FanElectricMap' object). If no scaling is wanted then "ign" should be entered.

"As Used" Fan Moment of Inertia

This is the new fan inertia that will override the value found in the 'FanSpecs' reference object. It can either be entered as a value, or "def". Where "def" will estimate the new fan inertia from the following equation using the new fan blade diameter, above, and the original fan blade diameter and inertia:

$$Inertia_{new} = Inertia_{old} * \left(\frac{D_{new}}{D_{old}} \right)^4$$

It is recommended that the new inertia, if known, be entered directly. If a new inertia is not to be used then "ign" should be entered.

Fan Boundary Type

☉ Speed Boundary from Mechanical Connection

Select this option if **Single Fan Map** or **Multiple Fan Map** was selected in the 'FanSpecs' reference object. An external 'Shaft' or 'SpeedBoundaryRot' connection to the 'Fan' is required with this option.

(☉) Normalized Fan Blade Position

For fans with a fixed geometry, this value should be set to "ign". If a fan design allows for varying the position (or angle) of the blades, this attribute controls that position (normalized). If a value or dependency object is specified, the solver will interpolate linearly between the two closest fan blade positions within the 'FanSpecs' reference object.

☉ Electric Fan (No

Select this option to model an electric fan if the **Electric Fan** option is





Mechanical Connection)	selected in the 'FanSpecs' reference object.
(☉) Input Signal	The input signal for the electric fan that will be imposed. Actuation of the input signal is also allowed.

Location

Location X (Global)	Specifies the absolute X location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a fan location of (0,0,0).
Location Y (Global)	Specifies the absolute Y location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a fan location of (0,0,0).
Location Z (Global)	Specifies the absolute Z location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a fan location of (0,0,0).

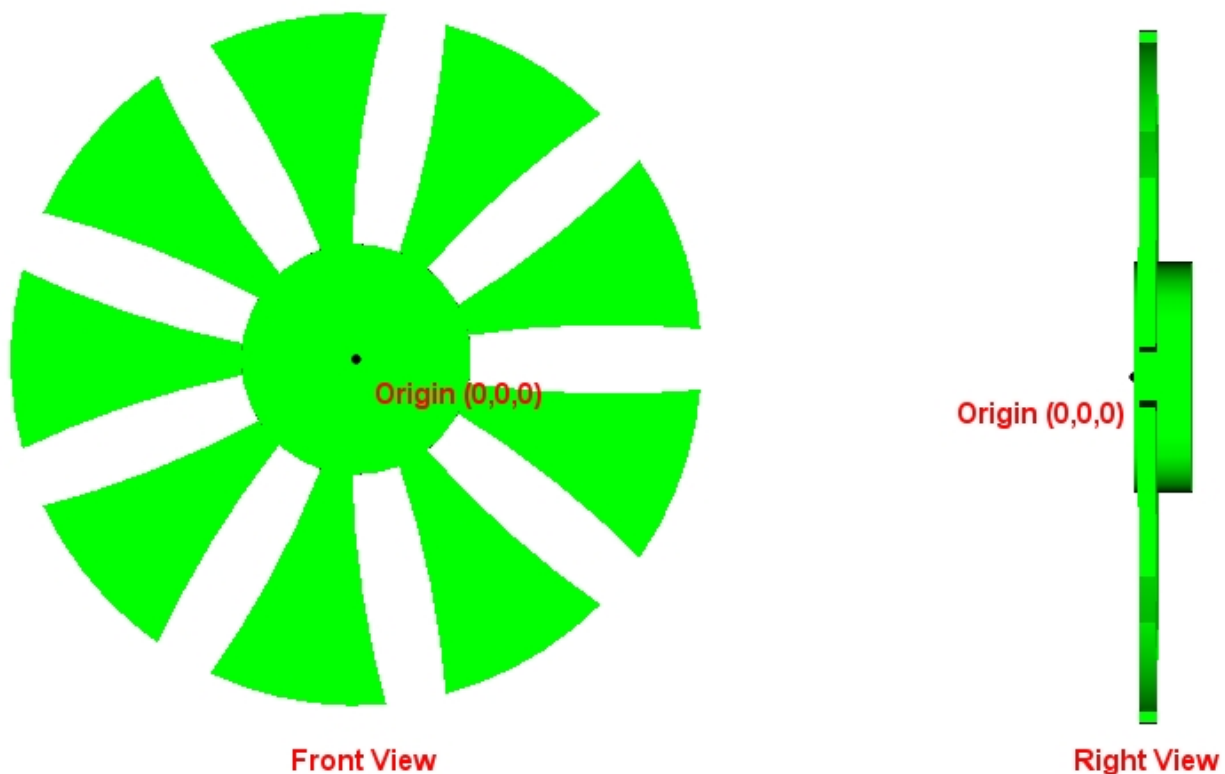


Figure 1: Location of a fan shroud in global space at location (0,0,0).



Options

Normalized Radial Distribution Reference

Name of the 'XYTableSimple' reference object to describe the radial distribution of the air flow through the fan. "ign" means no radial fan distribution will exist.

Flow Rate Multiplier

Multiplier of fan flow rate used to scale the whole map.

Efficiency Multiplier

Multiplier of fan efficiency used to scale the whole map.

Compressible Reference Conditions for Operating Point Correction

If the **Compressible Flow** option is selected in 'FanMap' or 'FanCoefficients', then certain plot and result (RLT) output of the fan involve correction of the simulation result to reference conditions such that it is relevant to compare to the corrected map input data or to other simulation results. These plots and RLT's usually have "Corrected" in the name. The following attributes control the reference conditions used for correction of simulation data within these plots/RLT's. These attributes will not change the raw simulation results, and will only impact the results displayed in these specific "corrected" plots/RLT's.

☉ Same as Map Object or External File

The pressure and temperature used for the corrected results will be identical to the conditions used in the original 'FanMap' object or external file. The reference conditions must be the same in each map or external file called by this object in order for this option to be used. This option is generally recommended so that comparisons to the map data are relevant.

☉ Enter Reference Conditions for Correction

Select this option to explicitly define the pressure and temperature to be used for the corrected output. This option is only needed if it is desirable to use different conditions than those used in the original 'FanMap' or external file (rare).

(☉) Reference Pressure

Pressure to which the mass flow rate will be corrected in the results.

(☉) Reference Temperature

Temperature to which the mass flow rate and speed will be corrected in the results.

Fan Model

Select the fan model to be used.

☉ Standard

Select this option to use the fan model included with GT-SUITE.

(☉) Time Constant (Explicit Solver Only)

A time constant is used to damp changes in mass flow rate to ensure that the solution remains stable (explicit solver only). Large values of the time constant, relative to the time step size, decrease the change in mass flow rate from time step to time step. The default option is available with a value that is internally calculated by the code. Note that if the user enters a value for the time constant that is smaller than the internally calculated value, the code will use the internally calculated value.

(☉) Relaxation Factor (Implicit Solver Only)

The relaxation factor may be used to help improve convergence of the implicit solver. Decreasing the relaxation factor slows changes in fan output (e.g. flow rate) from iteration to iteration to make the convergence more stable. Increasing the relaxation factor can speed up progress towards convergence, but can also introduce instability if increased too far. Values may range from 0.001 to 1. The "def" value is set to a fairly





conservative number ("def" = 0.025) so that most fans will converge without instability. Therefore it may be possible to speed up convergence by using a higher value.

A more robust option is the use of a 'XYTable' to specify a relaxation factor that varies dynamically as a function of the fan operating condition; a small factor is used when the fan is operating in a "flat" region on a fan map speed line, while a larger factor is used when in a "steep" region. If this option is chosen, it is recommended to use the pre-defined 'XYTable' called "RelaxFactor_Fan" which has been supplied in the GT-SUITE Library. If a custom table is created, it must be of the form $X = \text{slope} (1/L/\text{min})$, $Y = \text{relaxation factor}$.

☉ User

Select this option to point to a subroutine that was added to usercool.dll to model the fan behavior.

(☉) User Model Object Name

Name of a 'UserCodeReference' reference object that passes input data specified by the user to the user's subroutine.

Discretization

☉ Automatic Discretization

The fan will be discretized such that there are a maximum of 5 divisions in both the horizontal and vertical directions

☉ Horizontal Discretization Length

Specifies the target discretization length along the horizontal direction of the fan object. A value of "def" will assume a discretization length of 50 mm.

(☉) Vertical Discretization Length

Specifies the target discretization length along the vertical direction of the fan object. A value of "def" will assume a discretization length of 50 mm.

Fan Subassembly Port Number

The number entered here will be used as the subassembly port number for the mechanical connection of the fan in the external .gtm model created. If "def" is entered a number will be automatically assigned.

Parameterize Object to Case Setup

If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.

Part Name in GTM Export

The name given here will be used for the part name in the .gtm model that is created. When attempting to compare multiple objects the same part name should be given to directly compare those results in GT-POST. If "ign" is entered, the part name will be the object name.

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 object. The color choices include:

- | | |
|-------|---------|
| •Red | •Copper |
| •Blue | •Gold |





- Dark Blue

- Grey

- Green

- Black

Number of Blades

Number of fan blades in the fan model. This attribute is for visual representation only and does not affect results of the fan solution.

Plot Options

Transient Plot Frequency

A "snapshot" of all selected contour plots is taken at the specified interval. "ign" will cause a "snapshot" to be taken at the end of the simulation.





COOLFlowResistance

This template is used to model a flow resistance for the fluid where heat transfer will not be modeled.

Main

☉ Default Cross Section	Selecting this option will create a resistance plane that is the same size as the cross section of the 'COOLFlowSpace' at the current location.
☉ Rectangular Cross Section	Selecting this option will create a resistance that is of a specific (rectangular) dimension.
(☉) Height	Height of the resistance object.
(☉) Width	Width of the resistance object.
(☉) Depth	Depth of the resistance object.
Flow Resistance Model	Name of the 'FlowPDDropLossCoef', 'FlowPDDropPowerLaw', 'FlowPDDropTable', 'FlowPDDropTableRef', or 'FlowPDDropTempTable' reference object to describe the pressure drop of the resistance. If "def" is entered, a default orifice connection will be used.

Location

Component Origin Reference	<p>The origin reference location can be selected for the component from one of the following nine positions. The prefill is controlled from the attribute Definition of Component Origin in Tools > Options. All locations are assumed to be looking along the +X axis, with left being +Y and top being +Z.</p> <ul style="list-style-type: none"> • center: The center point of the resistance will be the origin. • upper right: The upper right corner of the resistance will be the origin. • lower right: The lower right corner of the resistance will be the origin. • upper left: The upper left corner of the resistance will be the origin. • lower left: The lower left corner of the resistance will be the origin. • top center: The top center point of the resistance will be the origin. • left center: The left center point of the resistance will be the origin. • bottom center: The bottom center point of the resistance will be the origin. • right center: The right center point of the resistance will be the origin.
-----------------------------------	--





Location X (Global)	Specifies the absolute X location of the component in the global coordinate system . The distance is measured from the global origin (0,0,0) to the center, front face of the object.
Location Y (Global)	Specifies the absolute Y location of the component in the global coordinate system . The distance is measured from the global origin (0,0,0) to the center, front face of the object.
Location Z (Global)	Specifies the absolute Z location of the component in the global coordinate system . The distance is measured from the global origin (0,0,0) to the center, front face of the object.

Discretization

Parameterize Object to Case Setup	If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.
--	---

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 object. The color choices include: <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green •Copper •Gold •Grey •Black





COOLFlowSpace

This template is used to model the flow space which will be used to constrain the flow in the model.

Cross Sections

Multiple columns should be used to represent an object that changes cross section. This is useful to create objects that are not extruded along a constant path.

Cross Section Name	Name of the 'CSBiRadial', 'CSCircle', 'CSCustom', 'CSEllipse', 'CSRect', or 'CSRoundRect' reference object describing the geometry of the flow space.
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. If wanted, "def" or "ign" can be entered for the last cross section.
Relative Y Location of Center	The relative distance in the Y-direction from the center of the first cross section to current cross section. "def" will have a value of 0.
Relative Z Location of Center	The relative distance in the Z-direction from the center of the first cross section to current cross section. "def" will have a value of 0.

Location

Location X (Global)	Specifies the absolute X location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if 'CSCustom' is used for the Cross Section Name . Figure 1 shown below has a flow space location of (0,0,0).
Location Y (Global)	Specifies the absolute Y location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if 'CSCustom' is used for the Cross Section Name . Figure 1 shown below has a flow space location of (0,0,0).
Location Z (Global)	Specifies the absolute Z location of the component's first cross section in the global coordinate system . The distance is measured from the global origin to the center, front face of the object, or the cross section local origin if 'CSCustom' is used for the Cross Section Name . Figure 1 shown below has a flow space location of (0,0,0).



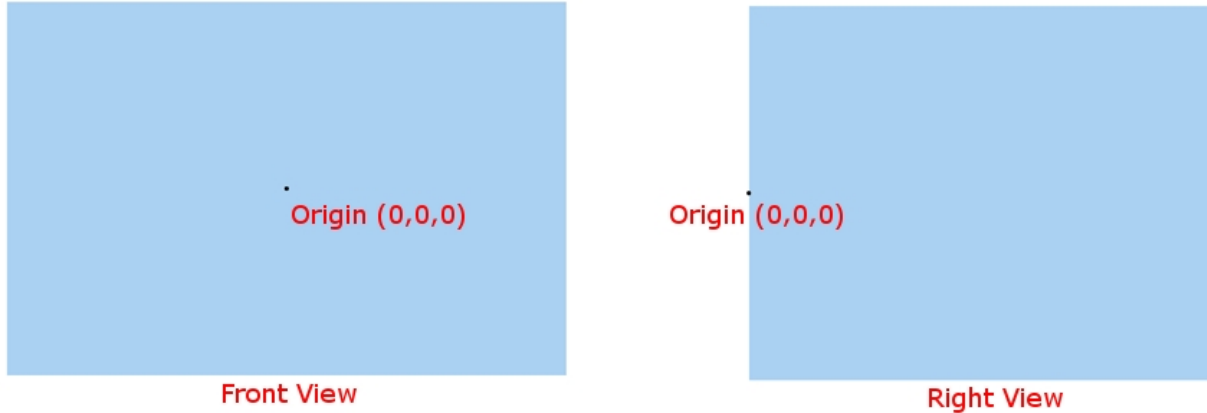


Figure 1: Location of a flow space in global space at location (0,0,0).

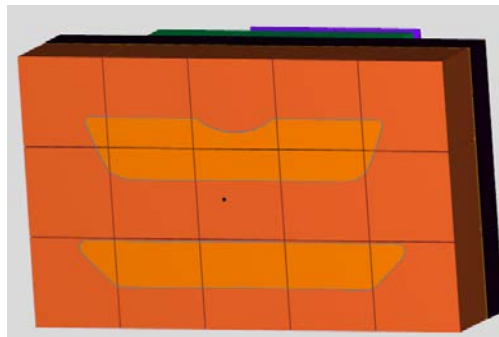
Initial State

Initial State Name Name of the 'FluidInitialState' reference object to describe the initial conditions inside the flow space. This does not include the initial conditions inside any heat exchangers.

Discretization

© Automatic Discretization

The air within the flow space will be divided using a target discretization length in both the Y and Z directions equal to 20% of the maximum dimension of the flow space (considering both Y and Z directions). This will result in 5 subvolumes in the direction of the largest dimension (width in the screenshot below).



Discretization in the X direction will only occur at the locations of components (equivalent to setting the **Discretization Length Along X direction** to "ign"). This option should generally provide a good balance between CPU time and model resolution.

© Discretization Length Along X direction

Specifies the target discretization length along the X direction of the flow space. A value of "def" will assume a discretization length of 50 mm. A value of "ign" will only look at the X location of the objects in the flow space for the creation of the layers. Additional layers will not be created.





(◎) Discretization Length Along Y direction	Specifies the target discretization length along the Y direction of the flow space. A value of "def" will assume a discretization length of 50 mm.
(◎) Discretization Length Along Z direction	Specifies the target discretization length along the Z direction of the flow space. A value of "def" will assume a discretization length of 50 mm.
Lateral Porosity Ratio	Specifies the porosity of the air grid along the Y and Z direction with 0 being non-porous and 1 being fully porous. The value entered will be passed to GT-SUITE as the forward and reverse discharge coefficient for the orifice connections to the lateral flowsplits. A value of "def" will automatically calculate the discharge coefficient based upon the neighboring geometry.
Flow Split Acceptance Ratio	Specifies the ratio of a cube's volume that must be contained inside the flow space for that flowsplit to be retained. If a lower ratio of a cube's volume is contained in the flow space, then COOL3D will throw out (disregard) that flowsplit when discretizing the flow space. The default value is 0.1. This attribute should be used to retain normally thrown out flowsplits that would significantly change the flow path inside the flow space. To illustrate, assume that a series of flowsplits are thrown out in a small section of the flow space. If this small section of the flow space represents a significant flow path that should be considered (i.e. a gap between two or more objects), then this attribute should be lowered so that those flowsplits are kept, thus retaining the significant flow path.
Parameterize Object to Case Setup	If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.

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 object. The color choices include: <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green •Copper •Gold •Grey •Black

Plot Options

Transient Plot Frequency	A "snapshot" of all selected contour plots is taken at the specified interval. "ign" will cause a "snapshot" to be taken at the end of the simulation.
---------------------------------	--





COOLFlowSpaceSimple

This template is used to model a simple flow space which will be used to constrain the flow in the model. The geometry of the flow space is automatically determined from the geometry of the components in the model.

Location

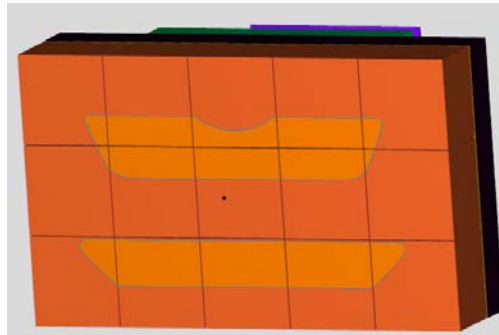
Distance (X) to 1st Component from Inlet	Specifies the location of the starting cross section for the flow space relative to the component located at the smallest X position.
Distance (X) after Last Component to Outlet	Specifies the location of the last cross section for the flow space relative to the component located at the largest X position.

Initial State

Initial State Name	Name of the 'FluidInitialState' reference object to describe the initial conditions inside the flow space. This does <u>not</u> include the initial conditions inside any heat exchangers.
---------------------------	--

Discretization

☉ Automatic Discretization	The air within the flow space will be divided using a target discretization length in both the Y and Z directions equal to 20% of the maximum dimension of the flow space (considering both Y and Z directions). This will result in 5 subvolumes in the direction of the largest dimension (width in the screenshot below).
-----------------------------------	--



Discretization in the X direction will only occur at the locations of components (equivalent to setting the **Discretization Length Along X direction** to "ign"). This option should generally provide a good balance between CPU time and model resolution.

☉ Discretization Length Along X direction	Specifies the target discretization length along the X direction of the flow space. A value of "def" will assume a discretization length of 50 mm. A value of "ign" will only look at the X location of the objects in the flow space for the creation of the layers. Additional layers will not be created.
(☉) Discretization Length Along Y direction	Specifies the target discretization length along the Y direction of the flow space. A value of "def" will assume a discretization length of 50 mm.





**(☉) Discretization
Length Along Z direction**

Specifies the target discretization length along the Z direction of the flow space. A value of "def" will assume a discretization length of 50 mm.

Lateral Porosity Ratio

Specifies the porosity of the air grid along the Y and Z direction with 0 being non-porous and 1 being fully porous. The value entered will be passed to GT-SUITE as the forward and reverse discharge coefficient for the orifice connections to the lateral flowsplits. A value of "def" will automatically calculate the discharge coefficient based upon the neighboring geometry.

**Flow Split Acceptance
Ratio**

Specifies the ratio of a cube's volume that must be contained inside the flow space for that flowsplit to be retained. If a lower ratio of a cube's volume is contained in the flow space, then COOL3D will throw out (disregard) that flowsplit when discretizing the flow space. The default value is 0.1.

This attribute should be used to retain normally thrown out flowsplits that would significantly change the flow path inside the flow space. To illustrate, assume that a series of flowsplits are thrown out in a small section of the flow space. If this small section of the flow space represents a significant flow path that should be considered (i.e. a gap between two or more objects), then this attribute should be lowered so that those flowsplits are kept, thus retaining the significant flow path.

**Parameterize Object to
Case Setup**

If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.

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 object. The color choices include:

- | | |
|------------|---------|
| •Red | •Copper |
| •Blue | •Gold |
| •Dark Blue | •Grey |
| •Green | •Black |

Plot Options

Transient Plot Frequency

A "snapshot" of all selected contour plots is taken at the specified interval. "ign" will cause a "snapshot" to be taken at the end of the simulation.





COOLHeatAddition

This object is used to model the pressure drop and heat input rate for a simplified tube/fin heat exchanger with a known heat transfer rate. The rate of heat input to the fluid may be specified either directly or as a reference object.

Main

"As Tested" Heat Addition Specifications Object	Name of the HeatAdditionSpecs reference object that describes the "as tested" heat addition configuration that was used in the lab to obtain the performance data (pressure drop).
Heat Input Rate	Heat transfer rate into the external fluid or the name of a dependency reference object.
External Initial State Name	Name of the FluidInitialState reference object to describe the initial conditions of the heat addition. If "def" is entered, the FluidInitialState in the COOLFlowSpace will be used.

Location

Component Origin Reference	<p>The origin reference location can be selected for the component from one of the following nine positions. The prefill is controlled from the attribute Definition of Component Origin in Tools > Options. All locations are assumed to be looking along the +X axis, with left being +Y and top being +Z.</p> <ul style="list-style-type: none"> • center: The center point of the resistance will be the origin. • upper right: The upper right corner of the resistance will be the origin. • lower right: The lower right corner of the resistance will be the origin. • upper left: The upper left corner of the resistance will be the origin. • lower left: The lower left corner of the resistance will be the origin. • top center: The top center point of the resistance will be the origin. • left center: The left center point of the resistance will be the origin. • bottom center: The bottom center point of the resistance will be the origin. • right center: The right center point of the resistance will be the origin.
-----------------------------------	--

Location X (Global)	Specifies the absolute X location of the component's first cross section in the global coordinate system . The distance is measured from the global
----------------------------	---





origin to the center, front face of the object. Figure 1 shown below has a heat addition location of (0,0,0).

Location Y (Global)

Specifies the absolute Y location of the component's first cross section in the [global coordinate system](#). The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a heat addition location of (0,0,0).

Location Z (Global)

Specifies the absolute Z location of the component's first cross section in the [global coordinate system](#). The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a heat addition location of (0,0,0).

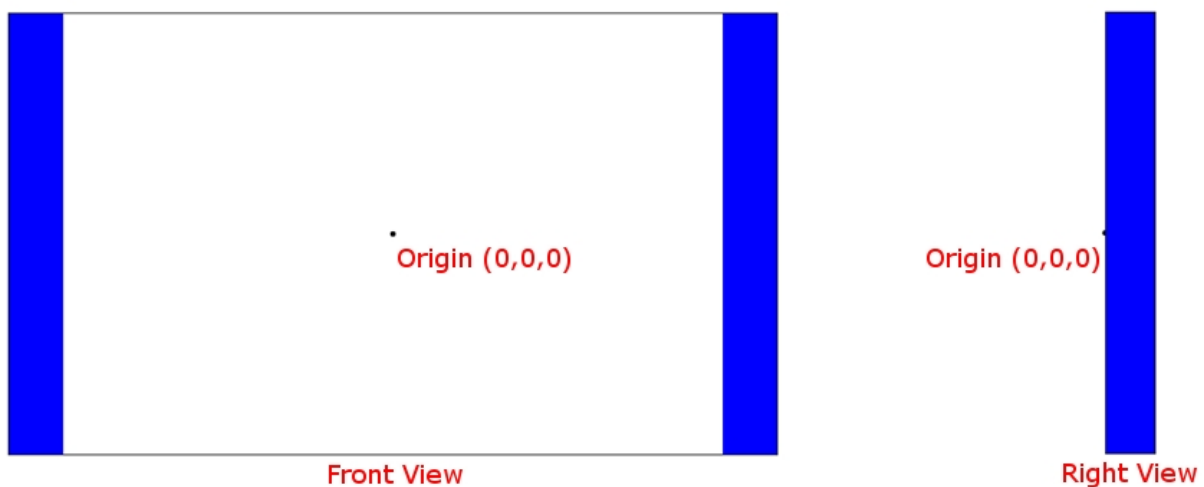


Figure 1: Location of a heat addition in global space at location (0,0,0).

"As Used" Configuration

Flow Orientation Override

Will override the **Flow Orientation** value defined in the ['HeatAdditionSpecs'](#) reference object.

- **ign:** No changes will be made to the orientation of the heat addition.
- **horizontal:** The heat addition will be considered at a horizontal orientation.
- **vertical:** The heat addition will be considered at a vertical orientation.

Anchor Point

The location chosen here will determine where the anchor point is placed for the scaling of the heat addition with **Change in Core Height** and **Change in Core Width**. All locations are assumed to be looking along the +X axis, with left being +Y and top being +Z.

- **center:** The center point of the heat addition will be fixed.





- **upper right:** The upper right corner of the heat addition will be fixed.
- **lower right:** The lower right corner of the heat addition will be fixed.
- **upper left:** The upper left corner of the heat addition will be fixed.
- **lower left:** The lower left corner of the heat addition will be fixed.
- **top center:** The top center point of the heat addition will be fixed.
- **left center:** The left center point of the heat addition will be fixed.
- **bottom center:** The bottom center point of the heat addition will be fixed
- **right center:** The right center point of the heat addition will be fixed.

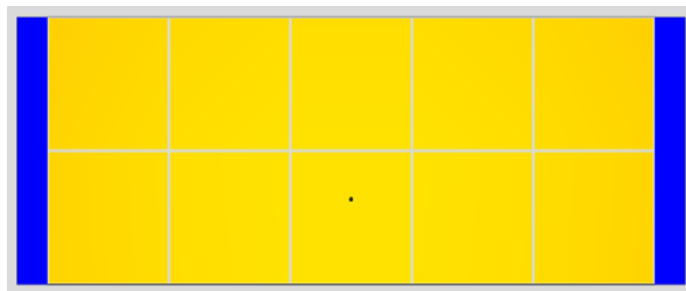
"As Used" Heat Addition Geometry Object (Scaling)

The name of an '[HAScaleGeomTubeFin](#)' reference object that is used to scale the geometry of the heat addition. These new dimensions will be used to predict the pressure drop of the heat addition. If scaling is not desirable then "ign" should be entered.

Discretization

☉ Automatic Discretization

The heat addition will be divided using a target discretization length in both the horizontal and vertical directions equal to 20% of the maximum dimension of the heat addition (considering both Y and Z directions). This will result in 5 subvolumes in the direction of the largest dimension (horizontal in the screenshot below).



This option should generally provide a good balance between CPU time and model resolution.

☉ Horizontal Discretization Length

Specifies the target discretization length along the horizontal direction of the heat addition object. A value of "def" will assume a discretization length of 50 mm.

(☉) Vertical

Specifies the target discretization length along the vertical direction of





Discretization Length	the heat addition object. A value of "def" will assume a discretization length of 50 mm.
Parameterize Object to Case Setup	If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.
Part Name in GTM Export	The name given here will be used for the part name in the .gtm model that is created. When attempting to compare multiple objects the same part name should be given to directly compare those results in GT-POST. If "ign" is entered, the part name will be the object name.

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 object. The color choices include: <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green •Copper •Gold •Grey •Black

Plot Options

Transient Plot Frequency	A "snapshot" of all selected contour plots is taken at the specified interval. "ign" will cause a "snapshot" to be taken at the end of the simulation.
---------------------------------	--





COOLHeatExchanger

This object is used to model the heat transfer between a fluid and the wall of a tube/fin heat exchanger. This object is configured such that the internal and external sides of the heat exchanger are modeled in detail based upon the geometry of the entered.

Main

"As Tested" Heat Exchanger Specifications Object	Name of the ' HeatExchangerSpecs ' reference object that describes the "as tested" heat exchanger configuration that was used in the lab to obtain the performance data.
Internal Initial State Name	Name of the ' FluidInitialState ' or ' RefrigInitialState ' reference object to describe the initial condition of the heat exchanger internal flow.
External Initial State Name	Name of the ' FluidInitialState ' reference object to describe the initial condition of the heat exchanger external flow. If "def" is entered, the ' FluidInitialState ' in the ' COOLFlowSpace ' will be used.
Initial Wall Temperature	Wall temperature at the start of the simulation.

Location

Component Origin Reference	<p>The origin reference location can be selected for the component from one of the following nine positions. The prefill is controlled from the attribute Definition of Component Origin in Tools > Options. All locations are assumed to be looking along the +X axis, with left being +Y and top being +Z.</p> <ul style="list-style-type: none"> • center: The center point of the resistance will be the origin. • upper right: The upper right corner of the resistance will be the origin. • lower right: The lower right corner of the resistance will be the origin. • upper left: The upper left corner of the resistance will be the origin. • lower left: The lower left corner of the resistance will be the origin. • top center: The top center point of the resistance will be the origin. • left center: The left center point of the resistance will be the origin. • bottom center: The bottom center point of the resistance will be the origin. • right center: The right center point of the resistance will be the origin.
-----------------------------------	--





- Location X (Global)** Specifies the absolute X location of the component's first cross section in the [global coordinate system](#). The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a heat exchanger location of (0,0,0).
- Location Y (Global)** Specifies the absolute Y location of the component's first cross section in the [global coordinate system](#). The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a heat exchanger location of (0,0,0).
- Location Z (Global)** Specifies the absolute Z location of the component's first cross section in the [global coordinate system](#). The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a heat exchanger location of (0,0,0).

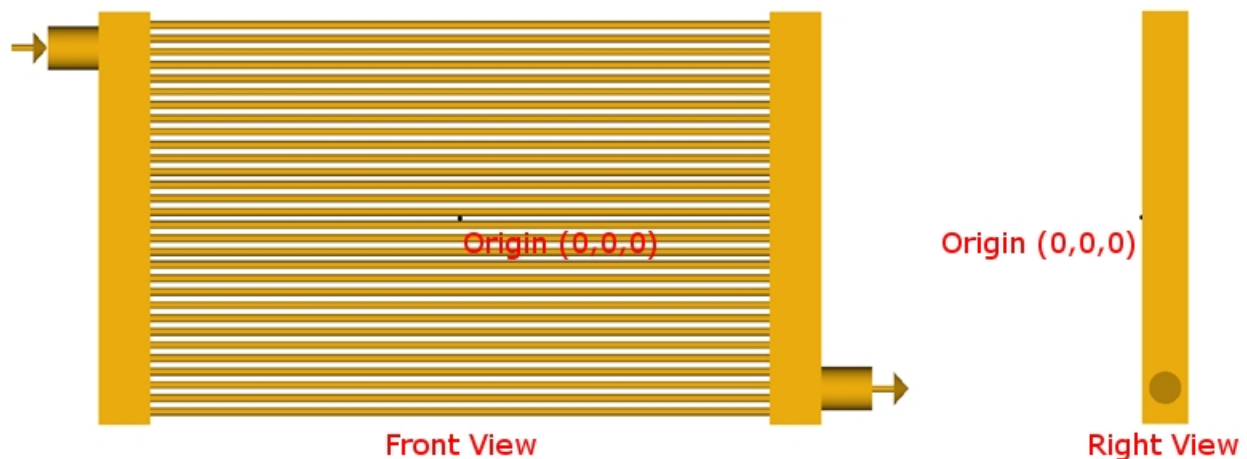


Figure 1: Location of a heat exchanger in global space at location (0,0,0).

"As Used" Configuration

This folder is used to scale the heat exchanger from the test data conditions.

"As Used" Heat Exchanger Geometry Object (Scaling)

The name of an ['HxScaleGeomTubeFin'](#) reference object that is used to scale the geometry of the heat exchanger. These new dimensions will be used to predict the performance of the heat exchanger. If scaling is not desirable then "ign" should be entered.

Performance Map Plots for "As Used" Geometry

Check this attribute to display the predicted performance plots for the "As Used" heat exchanger geometry. Plots for the Q/ETD and Pressure Drop will be automatically generated using the information found in ['HxNuMap'](#) and/or ['FlowPDrop'](#) of the "As Tested" input data. If these objects were not used for the "As Tested" input data, then predicted performance plots will not appear for the "As Used" geometry.

The predicted performance plots that are generated will be assigned to





Location of 1st Pass Relative to Center of Heat Exchanger Override (Single Core Only)

the heat exchanger part name in GT-POST. This will allow a unique plot to be created for each case if the scaled geometry is parameterized.

Will override the **Location of 1st Pass Relative to Center of Heat Exchanger** value defined in the ['HeatExchangerSpecs'](#) reference object.

- **ign:** No changes will be made to the location of the first pass.
- **Positive:** The first pass of the heat exchanger will be overridden to be at the most positive position.
- **Negative:** The first pass of the heat exchanger will be overridden to be at the most negative position.

This attribute will only have an effect if a single core heat exchanger is modeled. This attribute is unused if a multi-core heat exchanger is modeled.

Flow Orientation Override

Will override the **Flow Orientation** value defined in the ['HeatExchangerSpecs'](#) reference object.

- **ign:** No changes will be made to the orientation of the heat exchanger.
- **horizontal:** The heat exchanger will be considered at a horizontal orientation.
- **vertical:** The heat exchanger will be considered at a vertical orientation.

Inner Flow Direction of 1st Pass Override

Will override the **Inner Flow Direction of 1st Pass** value defined in the ['HeatExchangerSpecs'](#) reference object.

- **ign:** No changes will be made to the inner flow direction.
- **Positive:** The flow direction of the first pass will be overridden to flow along the positive axis.
- **Negative:** The flow direction of the first pass will be overridden to flow along the negative axis.

Anchor Point

The location chosen here will determine where the anchor point is placed for the scaling of the heat exchanger with **Change in Tube Length** and **Change in Number of Tubes (per Pass)**. All locations are assumed to be looking along the +X axis, with left being +Y and top being +Z.

- **center:** The center of the heat exchanger will be fixed.
- **upper right:** The upper right corner of the heat exchanger will be fixed.
- **lower right:** The lower right corner of the heat exchanger will be fixed.
- **upper left:** The upper left corner of the heat exchanger will be fixed.
- **lower left:** The lower left corner of the heat exchanger will be fixed.





fixed.

- **top center:** The top center point of the heat exchanger will be fixed.
- **left center:** The left center point of the heat exchanger will be fixed.
- **bottom center:** The bottom center point of the heat exchanger will be fixed
- **right center:** The right center point of the heat exchanger will be fixed.

Options

Internal Condense/Evaporate Water Vapor (Non- Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water for the internal fluid 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 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.
- **default:** Condensation and evaporation modeling is determined by the equivalent attribute in the associated ['HeatExchangerSpecs'](#) object. When that attribute is checked on, the **on_gas** option is used. Otherwise, this is set to **off**.

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.

External Condense/Evaporate Water Vapor (Non- Refrigerant Circuits)

Option to condense water vapor and/or evaporate liquid water for the external fluid 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 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.
- **default:** Condensation and evaporation modeling is determined by the equivalent attribute in the associated 'HeatExchangerSpecs' object. When that attribute is checked on, the **on_gas** option is used. Otherwise, this is set to **off**.

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.

☒ Pipe Length for Inlet and Outlet Volumes

Enabling this checkbox will allow flow volumes to be created at the inlet and outlet of the internal side of the heat exchanger that use the reference diameter entered in the heat exchanger geometry and the pipe length that is defined. This will allow the neighboring pipe geometry seen on the map to not influence the heat exchanger pressure drop and heat transfer performance in order to better understand the influences of area changes.

Receiver Dryer Mixing Threshold

Defines the liquid volume fraction (LVF) of all modeled Receiver/Dryers at which liquid/gas mixing will begin. When the LVF is greater than this threshold, liquid is output from the Receiver/Dryer components. When the LVF drops below this threshold, the outlet is assumed to be a mixture of the phases present in the '[ReceiverDryerRefrig](#)' component. ("def" = 0.1)

Internal Friction Multiplier

This multiplier is applied to the friction portion of the heat exchanger pressure losses. The multiplier only applies to heat exchanger components which are using the FlowPDRop reference object for the pressure losses, with the "Calculate Pressure Drop with Friction and Discharge Coefficient" option selected. Other pressure loss reference objects will not be affected by this multiplier. The "def" option is available and equal to a multiplier of 1.

Internal Heat Transfer Multiplier

This multiplier affects the heat transfer coefficient on the specified side of the heat exchanger. It applies to coefficients determined from the HxNuMap reference object. Because the total thermal resistance of the heat exchanger is dependent on both sides of the heat exchanger, using a multiplier of 2 will not necessarily double the heat transfer rate to the heat exchanger wall. The "def" option is available and equal to a multiplier of 1.

External Friction Multiplier

This multiplier is applied to the friction portion of the heat exchanger pressure losses. The multiplier only applies to heat exchanger





components which are using the FlowPDDrop reference object for the pressure losses, with the "Calculate Pressure Drop with Friction and Discharge Coefficient" option selected. Other pressure loss reference objects will not be affected by this multiplier. The "def" option is available and equal to a multiplier of 1.

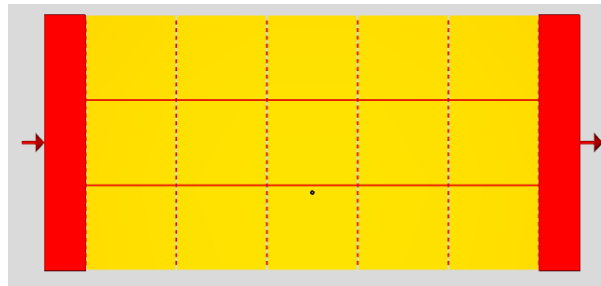
External Heat Transfer Multiplier

This multiplier affects the heat transfer coefficient on the specified side of the heat exchanger. It applies to coefficients determined from the HxNuMap reference object. Because the total thermal resistance of the heat exchanger is dependent on both sides of the heat exchanger, using a multiplier of 2 will not necessarily double the heat transfer rate to the heat exchanger wall. The "def" option is available and equal to a multiplier of 1.

Discretization

☉ Automatic Discretization

The heat exchanger will be divided using a target discretization length in both the axial and tangential directions equal to 20% of the maximum dimension of the heat exchanger (considering both Y and Z directions). This will result in 5 subvolumes in the direction of the largest dimension (axial in the screenshot below).



This option should generally provide a good balance between CPU time and model resolution.

☉ Axial (Flow Direction) Discretization Length

Specifies the target discretization length along the axial flow direction of the heat exchanger object. The axial flow direction is defined as the direction parallel to the internal flow path. A value of "def" will assume a discretization length of 50 mm.

(☉) Tangential Discretization Length

Specifies the target discretization length along the tangential flow direction of the heat exchanger object. The tangential flow direction is defined as the direction perpendicular to the internal flow path (not along the heat exchanger depth). A value of "def" will assume a discretization length of 50 mm.

Inlet (Internal) Subassembly Port Number

The number entered here will be used as the subassembly port number for the inlet flow of the heat exchanger in the external .gtm model created. If "def" is entered a number will be automatically assigned.

Outlet (Internal) Subassembly Port Number

The number entered here will be used as the subassembly port number for the outlet flow of the heat exchanger in the external .gtm model created. If "def" is entered a number will be automatically assigned.





Parameterize Object to Case Setup	If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.
Part Name in GTM Export	The name given here will be used for the part name in the .gtm model that is created. When attempting to compare multiple objects the same part name should be given to directly compare those results in GT-POST. If "ign" is entered, the part name will be the object name.

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 object. The color choices include: <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green •Copper •Gold •Grey •Black
<input checked="" type="checkbox"/> Internal Flow Direction Transparency Percent	Indicates the transparency level used when drawing the arrows that represent the internal flow direction. 0 indicates opaque (solid) and 90 indicates almost completely transparent.
<input checked="" type="checkbox"/> Internal Flow Direction Display Color	Indicates the color used when drawing the arrows that represent the internal flow direction. The color choices include: <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green •Copper •Gold •Grey •Black •Blue-Red •Red-Blue

Plot Options

Transient Plot Frequency	A "snapshot" of all selected contour plots is taken at the specified interval. "ign" will cause a "snapshot" to be taken at the end of the simulation.
---------------------------------	--





COOLSolidBlockage

This object is used to model the flow resistance of an object impeding flow in a flow space. It is created from a Solid Shape, using the imported solid geometry as the exact shape of the blockage. Holes cannot be added to the COOLSolidBlockage, they must be present in the imported geometry.

Discretization

Parameterize Object to Case Setup

If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.

Visual

Transparency Percent

Indicates the transparency level used when drawing the part. A value of 0 indicates opaque (solid) and a value of 90 indicates almost completely transparent.

Display Color

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

- | | |
|------------|---------|
| •Red | •Copper |
| •Blue | •Gold |
| •Dark Blue | •Grey |
| •Green | •Black |





COOLSolidFlowSpace

This template is used to model the flow space. This is required in order to create any other objects in COOL3D. It is created from a Solid Shape, using the imported solid geometry as the exact shape of the flowspace.

Location

Location X (Global)	Specifies the absolute X location of the component in the global coordinate system . The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a flow space location of (0,0,0).
Location Y (Global)	Specifies the absolute Y location of the component in the global coordinate system . The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a flow space location of (0,0,0).
Location Z (Global)	Specifies the absolute Z location of the component in the global coordinate system . The distance is measured from the global origin to the center, front face of the object. Figure 1 shown below has a flow space location of (0,0,0).

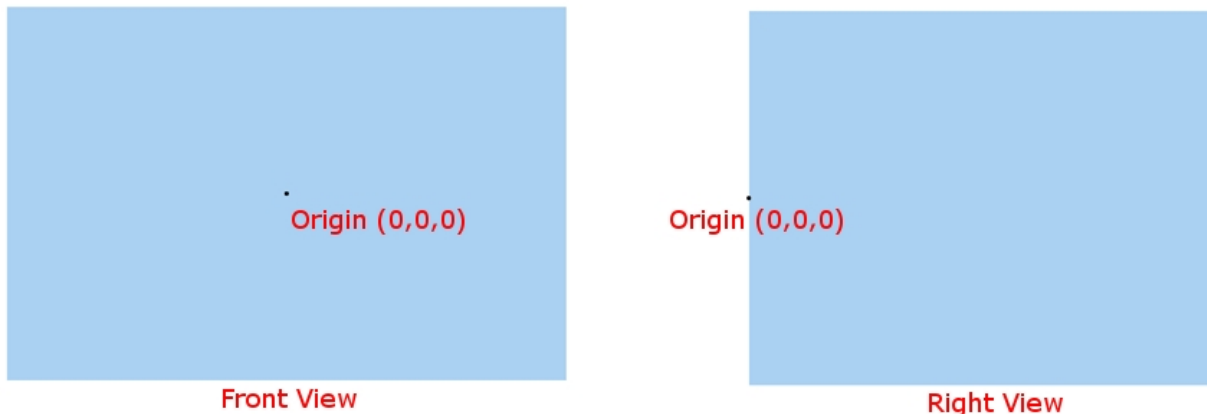


Figure 1: Location of a flow space in global space at location (0,0,0).

Main

Initial State Name	Name of the 'FluidInitialState' reference object to describe the initial conditions inside the flow space. This does <u>not</u> include the initial conditions inside the heat exchangers.
---------------------------	--

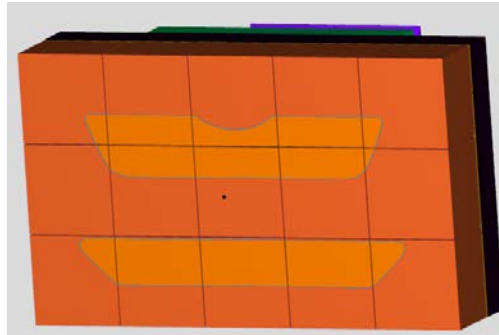
Discretization

☑ Automatic Discretization	The air within the flow space will be divided using a target discretization length in both the Y and Z directions equal to 20% of the maximum dimension of the flow space (considering both Y and Z directions). This
-----------------------------------	---





will result in 5 subvolumes in the direction of the largest dimension (width in the screenshot below).



Discretization in the X direction will only occur at the locations of components (equivalent to setting the **Discretization Length Along X direction** to "ign"). This option should generally provide a good balance between CPU time and model resolution.

© **Discretization Length Along X direction**

Specifies the target discretization length along the X direction of the flow space. A value of "def" will assume a discretization length of 50 mm. A value of "ign" will only look at the X location of the objects in the flow space for the creation of the layers. Additional layers will not be created.

(©) **Discretization Length Along Y direction**

Specifies the target discretization length along the Y direction of the flow space. A value of "def" will assume a discretization length of 50 mm.

(©) **Discretization Length Along Z direction**

Specifies the target discretization length along the Z direction of the flow space. A value of "def" will assume a discretization length of 50 mm.

Lateral porosity ratio

Specifies the porosity of the air grid along the Y and Z direction with 0 being non-porous and 1 being fully porous. The value entered will be passed to GT-SUITE as the forward and reverse discharge coefficient for the orifice connections to the lateral flowsplits. A value of "def" will automatically calculate the discharge coefficient based upon the neighboring geometry.

Flowsplit acceptance ratio

Specifies the ratio of a cube's volume that must be contained inside the flow space for that flowsplit to be retained. If a lower ratio of a cube's volume is contained in the flow space, then COOL3D will throw out (disregard) that flowsplit when discretizing the flow space. The default value is 0.1.

This attribute should be used to retain normally thrown out flowsplits that would significantly change the flow path inside the flow space. To illustrate, assume that a series of flowsplits are thrown out in a small section of the flow space. If this small section of the flow space represents a significant flow path that should be considered (i.e. a gap between two or more objects), then this attribute should be lowered so that those flowsplits are kept, thus retaining the significant flow path.

Parameterize Object to Case Setup

If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.





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 object. The color choices include:

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

Plot Options

Transient Plot Frequency A "snapshot" of all selected contour plots is taken at the specified interval. "ign" will cause a "snapshot" to be taken at the end of the simulation.





DiscretizationPlane

This template is used to build discretization planes. Discretization planes are defined as the plane perpendicular to a normal vector. The normal vector is given by 3 location attributes and 3 direction attributes.

Location

Location X (Global)	Specifies the absolute X location of the component in the global coordinate system . The distance is measured from the global origin to the center, front face of the object.
Location Y (Global)	Specifies the absolute Y location of the component in the global coordinate system . The distance is measured from the global origin to the center, front face of the object.
Location Z (Global)	Specifies the absolute Z location of the component in the global coordinate system . The distance is measured from the global origin to the center, front face of the object.
Direction X	The X-component of the normal vector describing the discretization plane.
Direction Y	The Y-component of the normal vector describing the discretization plane.
Direction Z	The Z-component of the normal vector describing the discretization plane.

Discretization

Parameterize Object to Case Setup	If checked, this object will become a parameter in Case Setup where it can be selected to be included or removed from Case to Case.
--	---

Visual

Discretization Plane Size	Specifies the length and width to be used when drawing the discretization plane in the graphical window.
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 object. The color choices include: <div style="display: flex; justify-content: space-between; margin-top: 10px;"> <div> <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green </div> <div> <ul style="list-style-type: none"> •Copper •Gold •Grey •Black </div> </div>





GEMMeshShape – General Mesh Shape

This template is used to represent a shape imported from an external geometry file (STL). 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 COOL3D component using the [Convert Mesh](#) 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](#).

Visual

Transparency Percent	Indicates the transparency level used when drawing the component. 0 indicates opaque (solid) and 90 indicates almost completely transparent.
Display Color	Indicates the color used when drawing the component. The color choices include: <ul style="list-style-type: none">• Red• Blue• Dark Blue• Green• Copper• Gold• Grey• Black





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 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](#). Solid shapes can also be converted into [Mesh Shapes](#) using the [Convert Solid to Mesh](#) operation.

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: <ul style="list-style-type: none">• Red• Blue• Dark Blue• Green• Copper• Gold• Grey• Black



CHAPTER 5: COOL3D Features

The COOL3D features section contains information on each of the features used in COOL3D. A description of each feature 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 COOL3D by clicking the button in the upper left corner of each template. This button will have an image of the template as well as a 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)

This section only matters if the template is used in GEM3D. Otherwise, this section can be ignored for COOL3D models.

☉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.





FlowOpening

This template is used to build fluid boundary conditions, or connections to another fluid circuit. The opening is defined as a plane perpendicular to a normal vector. The normal vector is given by 3 location attributes and 3 direction attributes. The shape of the opening will be extruded onto the 'COOLFlowSpace', which will then proceed to cut a hole for the fluid boundary or connection.

Main

Boundary Conditions

☉ Imposed Boundary Condition Reference Object

Specify the type of boundary condition that will be assigned to the opening.

Select this option to specify the name of a 'COOLEndEnv', 'COOLEndEnvRam', or 'COOLEndFlowInlet' reference object used to impose the boundary condition on the 'COOLFlowSpace'.

☉ Subassembly Port Number

Select this option to create a dangling 'SubAssExternalConn' connection at the opening to integrate the model with another circuit. Entering an integer will create a connection number to assist in the correct connectivity of the discretized model. A value of "def" will automatically assign a number.

☉ Characteristic Length of Flowsplit at Opening

It is recommended to specify the length of the attached flow volume at the boundary location to automatically create a 'FlowSplitGeneral' that uses the correct geometry and connectivity, which is required to link the discretized .ghx model with another flow part. This will then allow 'Pipe*' objects to connect directly to the external subassembly that is created when a .ghx file is discretized. The 'FlowSplitGeneral' is necessary because it is used to distribute the flow at the boundary between a 1D flow volume part and the neighboring 'MatrixFlowSplit', which contains multiple flow connections.

The length that is entered will be used to calculate the volume and characteristic length of the automatically generated 'FlowSplitGeneral'. The prefill value of 5 mm was selected to reduce the impact of any extra volume added to the system, while attempting to take into consideration any time step limitation of the explicit solver. If using the implicit solver, the value entered should have little impact on the stability of the solution. However, if using the explicit solver the time step may be affected if the volume size generated is the limiting part in the flow circuit. In this case, the characteristic length that is entered should be increased while taking into account any extra volume that may be added to the flow circuit.

A value of "ign", which is not recommended, will require a manual 'FlowSplitGeneral' to be created outside of the discretized model in order to integrate an existing flow circuit at this opening. Some models may evolve to this state in order to preserve old results. Using a non-"ign" value allows the model to be built easier.

Boundary Flow Direction

Define the direction of flow as an inlet or outlet boundary condition. This will affect the arrow direction of the discretized model only.





Cross Section Definition

Specify the cross section shape and geometrical properties.

☉ Cross Section Name

Name of the unique 'CSBiRadial', 'CSCircle', 'CSCustom', 'CSEllipse', 'CSRect', or 'CSRoundRect' reference object describing the geometry of the flow opening.

If this option is selected, then the attributes in the **Location** folder will need to be defined as well.

☉ Location of Opening for Flow

Selecting this option will automatically create an opening for the boundary condition.

- **First Cross Section** - the opening will be placed at the same location and with the same geometry as the first cross section in the 'COOLFlowSpace'.
- **Last Cross Section** - the opening will be placed at the same location and with the same geometry as the last cross section in the 'COOLFlowSpace'.

If this option is selected, then the attributes in the **Location** folder do not need to be defined.

Expansion Area Ratio

The flow volumes (flow splits) inside of the flow space that contact the opening can optionally have their expansion diameter for the port connected through the opening to the boundary condition increased beyond what is typically calculated from the dimensions of the flow split. The expansion area of each flow split will be multiplied by the factor entered here ("def" = 1.0).

Discharge Coefficient

The value for the discharge coefficient at the opening location. If a value of 1 is entered, and the **Imposed Boundary Condition Reference Object** above is 'COOLEndFlowInlet', then the value entered here will be replaced with a value of "def". A value of "def" will have the coefficient automatically calculated using the geometry of the mating flow components.

This attribute is unused if **Characteristic Length of Flowsplit at Opening** (above) is selected and set to "ign".

Location

The attributes in this folder are unused (invisible) if the radio button **Location of Opening for Flow** is selected in the **Main** folder.

Location Definition

The location of the boundary opening can be specified as local or global coordinates.

- **Global** – the reference point is defined as the origin (0,0,0).
- **Local** – the reference points is defined as the center front face location of the parent 'COOLFlowSpace' or 'COOLFlowSpaceSimple' object. If the 'COOLFlowSpace' or 'COOLFlowSpaceSimple' object changes location, then the boundary opening will adjust as well.





Location X	Specifies the X location of the component as either a global or local location depending on the Location Definition attribute above. The distance is measured from the reference point to the center, front face of the opening.
Location Y	Specifies the Y location of the component as either a global or local location depending on the Location Definition attribute above. The distance is measured from the reference point to the center, front face of the opening.
Location Z	Specifies the Z location of the component as either a global or local location depending on the Location Definition attribute above. The distance is measured from the reference point to the center, front face of the opening.
Direction X	The X-component of the normal vector describing the flow opening.
Direction Y	The Y-component of the normal vector describing the flow opening.
Direction Z	The Z-component of the normal vector describing the flow opening.

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 object. The color choices include: <ul style="list-style-type: none"> •Red •Blue •Dark Blue •Green •Copper •Gold •Grey •Black
Display Original Cross Section	When checked, this will draw the original cross section in the graphical window. This should initially be used to ensure proper placement of the flow opening.



HoleDuct

This template is used to create a hole or flap in a 'COOLDuct' component. One of the more common applications for use is to create the hole or flap openings in a fan shroud. The cross section of the opening will be projected onto the surface of the 'COOLDuct' component.

Main

Cross Section Name	Name of the 'CSBiRadial', 'CSCircle', 'CSCustom', 'CSEllipse', 'CSRect', or 'CSRoundRect' reference object that will be used to create the hole.
Flow Resistance Model	Name of the 'EffAreaRestriction', 'FlowPDropLossCoef', 'FlowPDropPowerLaw', 'FlowPDropTable', 'FlowPDropTableRef', or 'FlowPDropTempTable' reference object to describe the pressure drop of the hole. A 'ValveActuLiftAreaCon', 'ValveActuLiftCdConn', or 'ValveCheckSimpleConn' reference object can be used instead if a flap is to be modeled. If "def" is entered, a default orifice connection will be used.
Location	Specify the location of the opening.
Location Definition	<p>The location of the opening can be specified as local or global coordinates.</p> <ul style="list-style-type: none"> • Global – the reference point is defined as the origin (0,0,0). • Local – the reference points is defined as the center front face location of the parent 'COOLDuct' object. If the 'COOLDuct' object changes location, then the opening will adjust as well.
Location X	Specifies the X location of the component as either a global or local location depending on the Location Definition attribute above. The distance is measured from the reference point to the center, front face of the opening.
Location Y	Specifies the Y location of the component as either a global or local location depending on the Location Definition attribute above. The distance is measured from the reference point to the center, front face of the opening.
Location Z	Specifies the Z location of the component as either a global or local location depending on the Location Definition attribute above. The distance is measured from the reference point to the center, front face of the opening.
Direction X	The X-component of the normal vector describing the flow opening.
Direction Y	The Y-component of the normal vector describing the flow opening.
Direction Z	The Z-component of the normal vector describing the flow opening.





HoleVolume

This template is used to create a hole that will include the fluid volume. In addition, options are available to define different flow resistance models at both the inlet and outlet of the hole. If flaps are to be modeled, it is recommended to use the ['HoleVolumeZero'](#) feature instead.

Main

Cross Section Name	Name of the 'CSBiRadial' , 'CSCircle' , 'CSCustom' , 'CSEllipse' , 'CSRect' , or 'CSRoundRect' reference object that will be used to create the extruded hole.
Relative Y Location of Center	The relative distance in the Y-direction from the blockage center to the center of the hole.
Relative Z Location of Center	The relative distance in the Z-direction from the blockage center to the center of the hole.
Upstream Flow Resistance Model	Name of the 'EffAreaRestriction' , 'FlowPDDropLossCoef' , 'FlowPDDropPowerLaw' , 'FlowPDDropTable' , 'FlowPDDropTableRef' , or 'FlowPDDropTempTable' reference object to describe the pressure drop upstream of the hole (entering the hole). If "def" is entered, a default orifice connection will be used.
Downstream Flow Resistance Model	Name of the 'EffAreaRestriction' , 'FlowPDDropLossCoef' , 'FlowPDDropPowerLaw' , 'FlowPDDropTable' , 'FlowPDDropTableRef' , or 'FlowPDDropTempTable' reference object to describe the pressure drop downstream of the hole (exiting the hole). If "def" is entered, a default orifice connection will be used.





HoleVolumeObject

This template is used to create a hole in a blockage that is taken up by some other COOL* object (heat exchanger, heat addition, or fan). This allows the object to fit perfectly inside of the blockage without any leakage paths around the component.

Main

COOL3D Object Name to Fill Hole

Name of a COOL* object (heat exchanger, heat addition, or fan) that will be used to create the hole in the blockage. The object selected will automatically fit inside of the blockage and take up the space of the hole.





HoleVolumeZero

This template is used to create a hole that will have no fluid volume. This is typically used to model holes in very thin objects where the volume that exists is not of concern, or to create flaps inside of a blockage.

Main

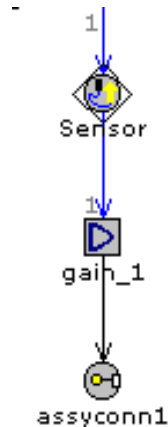
Cross Section Name	Name of the ' CSBiRadial ', ' CSCircle ', ' CSCustom ', ' CSEllipse ', ' CSRect ', or ' CSRoundRect ' reference object that will be used to create the hole.
Relative Y Location of Center	The relative distance in the Y-direction from the blockage center to the center of the hole.
Relative Z Location of Center	The relative distance in the Z-direction from the blockage center to the center of the hole.
Flow Resistance Model	Name of the ' EffAreaRestriction ', ' FlowPDDropLossCoef ', ' FlowPDDropPowerLaw ', ' FlowPDDropTable ', ' FlowPDDropTableRef ', or ' FlowPDDropTempTable ' reference object to describe the pressure drop of the hole. A ' ValveActuLiftAreaCon ', ' ValveActuLiftCdConn ', or ' ValveCheckSimpleConn ' reference object can be used instead if a flap is to be modeled. If "def" is entered, a default orifice connection will be used.





SensorConn3D - Sensor Connection

This template is used to sense quantities (pressure, temperature, velocity, etc) in a flow space 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.
Note: When selecting the Velocity (+X) or Velocity (-X) signal from the COOLFlowSpace object, the +X and -X denote the X-location for the subsequent layer that will be created. +X represents the most positive X-value (the outlet face of a flowsplit), and -X represents the most negative X-value (the inlet face of a flowsplit).

(only the attributes associated with the selected radio button need to be filled out)

☉ 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 subvolumes, 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 centroids, the centroid value will be reported. For example, if a given 'Pipe*' part has two subvolumes (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.

- **total:** 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.

© User-Defined Sensed Species Object

The name of the 'Fluid*' 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.

© Coverage or Combined Mixture Vapor Species Name

The text name of a surface coverage species used when sensing "Average Coverage" from a 'ChemConn' between a 'SurfaceReactions' part and an aftertreatment flow component. For example "Ce₂O₃". The name must be typed as text.

This attribute has the dual purpose of also supporting the sensing of combined mixture vapor species, the vapor of a 'FluidMixtureCombined'. 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.

Calculation for Sensed Quantity for Matrix* Part

The selected choice will determine the mathematical operation to be performed on the sensed quantity from the 'Matrix*' part. This option can only be used in COOL3D models.

- **average:** 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.
- **local:** If "local" is selected, then the quantity will be sensed from the local element found at the location entered in the **Main** folder.

Visual

Display Color

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

- | | |
|------------|---------|
| •Red | •Copper |
| •Blue | •Gold |
| •Dark Blue | •Grey |
| •Green | •Black |





Show Sensor Label

If checked, this will display the object 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:

τ	=	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).



CHAPTER 6: COOL3D Cross Sections

The COOL3D cross sections section contains information on each of the cross sections used in COOL3D. A description of each cross section 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 COOL3D by clicking the button in the upper left corner of each template. This button will have an image of the template as well as a small question mark symbol.



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. Cross sections are used to build flow space shapes, blockage shapes, create holes in blockages, and create an opening for a fan in a fan shroud. Multiple cross sections of different shapes can be combined to create complex component shapes.

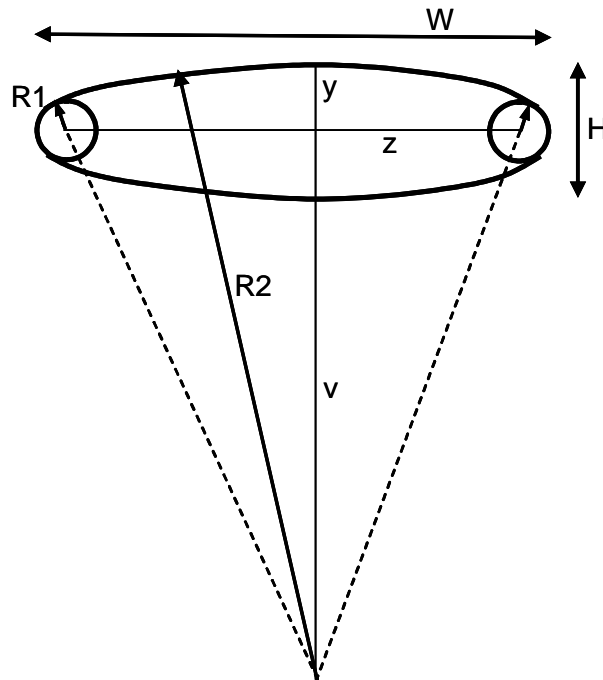
Geometry

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 may be set to "def" as long as the other 3 are specified and its value will be calculated automatically.

Radius 1 (Small radius)	Small radius to be used on the 2 sides of the shape (R_1). "def" is allowed (see note above).
Radius 2 (Large radius)	Large radius to be used on the top and bottom of the shape (R_2). "def" is allowed (see note above).
Height	Overall height of the shape (H). "def" is allowed (see note above).
Width	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 – Circle Cross Section

This template is used to model a circular cross section. Cross sections are used to build flow space shapes, blockage shapes, create holes in blockages, and create an opening for a fan in a fan shroud. 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 model an elliptical cross section. Cross sections are used to build flow space shapes, blockage shapes, create holes in blockages, and create an opening for a fan in a fan shroud. Multiple cross sections of different shapes can be combined to create complex component shapes.

Geometry

Major Diameter	Specifies the major diameter of the ellipse.
Minor Diameter	Specifies the minor diameter of the ellipse.



CSRect – Rectangular Cross Section

This template is used to model a rectangular cross section. Cross sections are used to build flow space shapes, blockage shapes, create holes in blockages, and create an opening for a fan in a fan shroud. Multiple cross sections of different shapes can be combined to create complex component shapes.

This template can also be referenced from 'CFDBoundary' to define the boundary that is used for the data mapping.

Geometry

Height	Specifies the height of the rectangle.
Width	Specifies the width of the rectangle.



CSRoundRect – Rounded Rectangular Cross Section

This template is used to model a rectangular cross section with rounded corners. Cross sections are used to build flow space shapes, blockage shapes, create holes in blockages, and create an opening for a fan in a fan shroud. Multiple cross sections of different shapes can be combined to create complex component shapes.

Geometry

Height	Specifies the height of the rounded rectangle.
Width	Specifies the width of the rounded rectangle.
Radius	Specifies the radius of the corners of the rounded rectangle.



CHAPTER 7: COOL3D Reference Templates

The COOL3D reference templates section contains information on each of the reference objects used in COOL3D. A description of each reference object 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 COOL3D by clicking the button in the upper left corner of each template. This button will have an image of the template as well as a small question mark symbol.





COOL3D EndEnv – Pressure Boundary

This template is a "dependency reference template", only available in COOL3D, that may be called to describe end environment boundary conditions of pressure, temperature, and composition.

Main

Pressure	<p>Absolute pressure in the environment or the name of a dependency reference object.</p> <p>One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a pressure contour in the boundary object.</p>
Temperature	<p>Temperature in the environment or the name of a dependency reference object.</p> <p>One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a temperature contour in the boundary object.</p>
<input checked="" type="checkbox"/> Temperature Shift	<p>Option to add a temperature offset, which can be specified as constant, time-varying, or a RLT dependence. This option is particularly useful if a variable temperature shift is needed and 'CFDBoundary' is used to specify temperature.</p>
Pressure Flag	<p>One of the following choices:</p> <ul style="list-style-type: none"> • standard(total) indicates that the pressure and temperature will be imposed as total (stagnation) conditions at the inlet of the attached flow component. This is the most physically-meaningful boundary condition and <u>should normally be used</u>. • inlet-static indicates that the pressure and temperature will be imposed as static conditions at the inlet of the attached flow component. This boundary condition should only be used when the flow is <u>into</u> the system. Because this option can introduce feedback into the flow solution, <u>it should not be used when the pressures just inside the boundary are fluctuating</u>, such as at the inlet manifold of an engine. This option is made available for steady flow tests, such as a simulated flow bench. • outlet-reversing indicates that the environment is primarily an outlet (exhaust) environment and that if the flow reverses back into the system from the environment, the source temperature will be kept at the last temperature that was exhausted. This feature is intended for exhaust systems where there are brief intermittent reversals of flow at the tailpipe outlet. These reversals should cause ingestion of the warm mixture of the same temperature just exhausted a moment before. This should be used when conducting an acoustic analysis of the exhaust system.
Composition	<p>Name of a 'Fluid*' reference object that defines the composition of the environment.</p>





Altitude and Humidity

Altitude

Check the box below to model changes in fluid properties caused by altitude changes.

☒ **Altitude**

If the simulation is to occur over a range of altitudes, the attributes **Altitude** and **Reference Altitude** can be specified to correct the pressure and temperature.

Altitude of the environment or the name of a dependency reference object for which the temperature and pressure should be corrected to. If the correction of temperature and pressure for altitude is not desired, this attribute should be set to "ign". ("def" = 0.0 = sea level)

☒ **Reference Altitude**

Altitude at which the 'COOL3D' pressure and temperature correction will refer to. ("def" = 0.0 = sea level)

☒ **Altitude Correction For...**

This attribute will be irrelevant if **Altitude** attribute above is set to "ign."

- **PressureOnly** indicates that only the 'COOL3D' pressure will be adjusted according to the equations below and that temperature is not adjusted due to the specified altitude. This option is appropriate when altitude-adjusted mean sea level pressures are used for the 'COOL3D'>Main>**Pressure** attribute and the local-elevation temperature is known. This is appropriate for many applications as altitude-adjusted temperatures are not commonly reported.
- **PresAndTemp** indicates that both the 'COOL3D' pressure and temperature will be adjusted due to altitude according to the equations below.
- **TemperatureOnly** indicates that only the 'COOL3D' temperature will be corrected according to the equations below and that pressure is not adjusted due to the specified altitude.
- **[ALTOPT]** creates a parameter that is used to set the options **PressureOnly**, **PresAndTemp**, and **TemperatureOnly** in a dropdown menu in Case Setup.

Humidity

**Relative Humidity
(Added to specified fresh
air Composition)**

Complete the attributes below to model humidity in a gas.

Relative humidity of Humidity Species (specified below) in the fresh air (i.e., excluding any burned gases and fuel in the mixture) evaluated at the initial environment temperature and pressure. If the environment pressure and/or temperature is transient, the added absolute humidity remains constant and the relative humidity of the fluid source will change. A value of "ign" results in no humidity correction to the composition.

- If dry air without any burned species is specified in the Composition attribute of the Main folder, then the Humidity Species corresponding to the entered humidity is added to the composition.
- If unburned H₂O is already specified in the Composition attribute, then only "ign" is allowed.





- If a mixture of dry air and burned species are specified in the Composition attribute, then Humidity Species will be added to produce the specified humidity for only the unburned species (i.e., dry air). This approach enables a simple way to impose the relative humidity of a mixture of fresh wet air and EGR.
- For liquid circuits, this attribute should always be set to "ign"

Recall that at a given temperature the maximum possible relative humidity is the ratio of pressure and the saturation pressure or 1,

$$\phi_{\max} = \min\left(P/P_{\text{sat}}|_T, 1\right)$$

A relative humidity greater than this maximum is not allowed. Note that a relative humidity specification should be avoided when the fluid temperature is near critical ($T_{\text{crit,H}_2\text{O}} = 647\text{K}$) because the humidity content becomes extremely sensitive to the relative humidity value (and undefined when supercritical). For these cases, H_2O can be added manually using a 'FluidMixture' reference object instead.

Humidity Species

'FluidGas', 'FluidNASA-LiqGas', or 'FluidPreDefined' reference object for H_2O vapor. Non- H_2O humidity species are not supported at this time. Parameters are allowed, and "ign" is permitted if and only if Relative Humidity is set to "ign".

☉ Apply Humidity to Initial Conditions

When choosing this option, pressure and temperature defined in the Main Folder are taken as the underlying reference condition for the above specified humidity value.

☉ Reference Pressure

The entered pressure is taken as the reference pressure for the above entered relative humidity. In case the entered value and the pressure defined in the Main Folder differ, the relative humidity is re-calculated to match the imposed fluid pressure. This implies that the absolute humidity is held constant after it is calculated using the reference pressure.

(☉) Reference Temperature

The entered temperature is taken as the reference temperature for the above entered relative humidity. In case the entered value and the temperature defined in the Main Folder differ, the relative humidity is re-calculated to match the imposed fluid temperature. This implies that the absolute humidity is held constant after it is calculated using the reference temperature.

Note: The altitude corrections for temperature and pressure are based on the following equations and are intended for air only (not applicable to other gases or liquids). The altitude corrections are limited to geometric altitudes at or below 86 km.

The geopotential altitude corrects for the variation in the acceleration due to gravity and is used only for internal calculations. The geopotential altitude is related to the geometric altitude by the following equation:





$$z = \frac{r_0 h}{r_0 + h}$$

where

z = the geopotential altitude

r_0 = the radius of the Earth = 6,356 km

h = the geometric altitude

The table below uses the geopotential altitude to lookup which altitude zone should be used.

Standard Atmosphere Model Altitude Zones

i	z_i , km'	L_i , K/km'
0	0	-6.5
1	11	0.0
2	20	1.0
3	32	2.8
4	47	0.0
5	51	-2.8
6	71	-2.0
7	84.852	--

Sea level conditions are $T_0 = 288.15$ K and $P_0 = 101,325$ Pa when **Reference Altitude** = 0 meters. Unless the **PressureOnly** option is selected, the corrected temperature at various altitudes is found using:

$$T = T_i + L_i(z - z_i)$$

where

T = corrected temperature

T_i = temperature at the start of the current altitude zone

L_i = temperature gradient of the current altitude zone

z = current geopotential altitude

z_i = geopotential altitude at the start of the current altitude zone

Unless the **TemperatureOnly** option is selected, the corrected pressure is found using one of two equations. If $L_i \neq 0$, the pressure is found using the equation:

$$P = P_i \left(\frac{T_i}{T} \right)^{\left(\frac{g_0 W_0}{R L_i} \right)}$$

where

P = corrected pressure

P_i = pressure at the start of the current altitude zone

g_0 = acceleration due to gravity, 9.80665 m/s²

W_0 = molar mass of air, 28.9644 kg/kmol

R = gas constant, 8,314.32 J/kmol-K





If $L_i = 0$, the pressure is calculated using:

$$P = P_i \exp\left(\frac{-g_0 W_0 (z - z_i)}{RT_i}\right)$$

References

Mattingly, J. (2006). Appendix A. In *Elements of propulsion: Gas turbines and rockets* (p. 789). Reston, Va.: American Institute of Aeronautics and Astronautics.





COOL3D EndEnvRam – End Environment with Ram Velocity

This template is a "dependency reference template", only available in COOL3D, that may be called to describe end environment boundary conditions of pressure, temperature, and composition when the effect of ram air velocity are to be taken into account. The total pressure at the inlet boundary will be calculated as follows from the data entered below:

$$P_{total} = P_{static} + \frac{1}{2} c_{pc} \rho V^2$$

- P_{total} = Total pressure in the environment
 P_{static} = **Static Pressure** (specified below)
 c_{pc} = **Ram Air Pressure Coefficient** (specified below)
 ρ = Density at the specified atmospheric temperature and pressure
 V = **External Ram Air Velocity** (specified below)

Main

Static Pressure	<p>Static pressure in the environment or the name of a dependency reference object.</p> <p>One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a pressure contour in the boundary object.</p>
Temperature	<p>Temperature in the environment or the name of a dependency reference object.</p> <p>One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a temperature contour in the boundary object.</p>
<input checked="" type="checkbox"/> Temperature Shift	<p>Option to add a temperature offset, which can be specified as constant, time-varying, or a RLT dependence. This option is particularly useful if a variable temperature shift is needed and 'CFDBoundary' is used to specify temperature.</p>
<input checked="" type="radio"/> External Ram Air Velocity	<p>Velocity of the ram air relative to the vehicle or the name of a dependency reference object.</p> <p>One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a velocity contour in the boundary object.</p>
<input checked="" type="radio"/> Vehicle Part Providing Ram Air Velocity	<p>The name of a 'VehicleBody' part in the model that will provide the velocity of ram air.</p>
Ram Air Pressure Coefficient	<p>Coefficient to the dynamic pressure term. Caution: For flow entering a system, this value should be a positive number between 1 and 0. For flow exiting a system, however, this value should be a negative number between 0 and -1.</p>
Composition	<p>Name of a 'Fluid*' reference object that defines the composition of the environment.</p>

Altitude and Humidity

Altitude	Check the box below to model changes in fluid properties cause by altitude changes.
-----------------	---





☒ **Altitude**

If the simulation is to occur over a range of altitudes, the attributes **Altitude** and **Reference Altitude** can be specified to correct the pressure and temperature.

Altitude of the environment or the name of a dependency reference object for which the temperature and pressure should be corrected to. If the correction of temperature and pressure for altitude is not desired, this attribute should be set to "ign". ("def" = 0.0 = sea level)

☒ **Reference Altitude**

Altitude at which the 'COOL3DEnvRam' pressure and temperature correction will refer to. ("def" = 0.0 = sea level)

☒ **Altitude Correction For...**

This attribute will be irrelevant if **Altitude** attribute above is set to "ign."

- **PressureOnly** indicates that only the 'COOL3DEnvRam' pressure will be adjusted according to the equations below and that temperature is not adjusted due to the specified altitude. This option is appropriate when altitude-adjusted mean sea level pressures are used for the 'COOL3DEnvRam'>Main>**Pressure** attribute and the local-elevation temperature is known. This is appropriate for many applications as altitude-adjusted temperatures are not commonly reported.
- **PresAndTemp** indicates that both the 'COOL3DEnvRam' pressure and temperature will be adjusted due to altitude according to the equations below.
- **TemperatureOnly** indicates that only the 'COOL3DEnvRam' temperature will be corrected according to the equations below and that pressure is not adjusted due to the specified altitude.
- **[ALTOPT]** creates a parameter that is used to set the options **PressureOnly**, **PresAndTemp**, and **TemperatureOnly** in a dropdown menu in Case Setup.

Humidity

**Relative Humidity
(Added to specified fresh
air Composition)**

There are two options available to specify the reference conditions for the relative humidity entered above.

Relative humidity of Humidity Species (specified below) in the fresh air (i.e., excluding any burned gases and fuel in the mixture) evaluated at the initial environment temperature and pressure. If the environment pressure and/or temperature is transient, the added absolute humidity remains constant and the relative humidity of the fluid source will change. A value of "ign" results in no humidity correction to the composition.

- If dry air without any burned species is specified in the Composition attribute of the Main folder, then the Humidity Species corresponding to the entered humidity is added to the composition.
- If unburned H₂O is already specified in the Composition attribute, then only "ign" is allowed.
- If a mixture of dry air and burned species are specified in the Composition attribute, then Humidity Species will be added to produce the specified humidity for only the unburned species (i.e.,





dry air). This approach enables a simple way to impose the relative humidity of a mixture of fresh wet air and EGR.

- For liquid circuits, this attribute should always be set to "ign"

Recall that at a given temperature the maximum possible relative humidity is the ratio of pressure and the saturation pressure or 1,

$$\phi_{\max} = \min\left(P/P_{\text{sat}}|_T, 1\right)$$

A relative humidity greater than this maximum is not allowed. Note that a relative humidity specification should be avoided when the fluid temperature is near critical ($T_{\text{crit,H}_2\text{O}} = 647\text{K}$) because the humidity content becomes extremely sensitive to the relative humidity value (and undefined when supercritical). For these cases, H_2O can be added manually using a 'FluidMixture' reference object instead.

Humidity Species

'FluidGas', 'FluidNASA-LiqGas', or 'FluidPreDefined' reference object for H_2O vapor. Non- H_2O humidity species are not supported at this time. Parameters are allowed, and "ign" is permitted if and only if Relative Humidity is set to "ign".

☉ Apply Humidity to Initial Conditions

When choosing this option, pressure and temperature defined in the Main Folder are taken as the underlying reference condition for the above specified humidity value.

☉ Reference Pressure

The entered pressure is taken as the reference pressure for the above entered relative humidity. In case the entered value and the pressure defined in the Main Folder differ, the relative humidity is re-calculated to match the imposed fluid pressure. This implies that the absolute humidity is held constant after it is calculated using the reference pressure.

(☉) Reference Temperature

The entered temperature is taken as the reference temperature for the above entered relative humidity. In case the entered value and the temperature defined in the Main Folder differ, the relative humidity is re-calculated to match the imposed fluid temperature. This implies that the absolute humidity is held constant after it is calculated using the reference temperature.

Note: The altitude corrections for temperature and pressure are based on the following equations and are intended for air only (not applicable to other gases or liquids). The altitude corrections are limited to geometric altitudes at or below 86 km.

The geopotential altitude corrects for the variation in the acceleration due to gravity and is used only for internal calculations. The geopotential altitude is related to the geometric altitude by the following equation:

$$z = \frac{r_0 h}{r_0 + h}$$





where

z = the geopotential altitude

r_0 = the radius of the Earth = 6,356 km

h = the geometric altitude

The table below uses the geopotential altitude to lookup which altitude zone should be used.

Standard Atmosphere Model Altitude Zones

i	z_i , km'	L_i , K/km'
0	0	-6.5
1	11	0.0
2	20	1.0
3	32	2.8
4	47	0.0
5	51	-2.8
6	71	-2.0
7	84.852	--

Sea level conditions are $T_0 = 288.15$ K and $P_0 = 101,325$ Pa when **Reference Altitude** = 0 meters. Unless the **PressureOnly** option is selected, the corrected temperature at various altitudes is found using:

$$T = T_i + L_i(z - z_i)$$

where

T = corrected temperature

T_i = temperature at the start of the current altitude zone

L_i = temperature gradient of the current altitude zone

z = current geopotential altitude

z_i = geopotential altitude at the start of the current altitude zone

Unless the **TemperatureOnly** option is selected, the corrected pressure is found using one of two equations. If $L_i \neq 0$, the pressure is found using the equation:

$$P = P_i \left(\frac{T_i}{T} \right)^{\left(\frac{g_0 W_0}{R L_i} \right)}$$

where

P = corrected pressure

P_i = pressure at the start of the current altitude zone

g_0 = acceleration due to gravity, 9.80665 m/s²

W_0 = molar mass of air, 28.9644 kg/kmol

R = gas constant, 8,314.32 J/kmol-K

If $L_i = 0$, the pressure is calculated using:





$$P = P_i \exp\left(\frac{-g_0 W_0 (z - z_i)}{RT_i}\right)$$

References

Mattingly, J. (2006). Appendix A. In *Elements of propulsion: Gas turbines and rockets* (p. 789). Reston, Va.: American Institute of Aeronautics and Astronautics.





COOL3D EndFlowInlet – Imposed Flow Rate

This template is a "dependency reference template", only available in COOL3D, that may be called to impose a flow rate into or out of an attached flow component. The flow rate of the fluid may be specified as volumetric flow rate, mass flow rate, velocity or mass flux. Positive values for volumetric flow rate, mass flow rate, velocity or mass flux indicate that flow is out of the 'COOL3D EndFlowInlet' part and into the system.

When this template is used, the code actually calculates the pressure at the boundary that is required to meet the flow rate specified by the user. This can cause stability problems if the user specifies a flow rate that is not appropriate for the system. For example, if a huge amount of flow is specified to go through a small pipe, orifice, valve, etc., then the pressure in the 'COOL3D EndFlowInlet' and any upstream flow components will rise very rapidly. Eventually, the density and/or Mach number will increase enough to meet the specified flow rate, but the resultant fluid state may be so ridiculous that the fluid properties, etc. can become unstable before the specified flow rate is met. The stability of the solution is particularly sensitive for liquids because the density changes very little as the pressure rises, resulting in extremely rapid pressure increases.

Main

☉ Volumetric Flow Rate

Volumetric flow rate to be imposed at the inlet of the attached flow component or the name of a dependency reference object.

Note that the pressure at the inlet will vary as necessary to achieve the specified volumetric flow rate. As a result, the density of the incoming air will vary, and therefore, this attribute cannot be used to impose a "standardized" volumetric flow rate. ("Standardized" refers to a volumetric flow rate that has been corrected to a reference temperature and pressure.) To impose a standardized volumetric flow rate, please refer to **Mass Flow Rate** immediately below.

☉ Mass Flow Rate / Air scfm

Mass flow rate to be imposed at the inlet of the attached flow component or the name of a dependency reference object.

If a "standardized" volumetric flow rate is to be imposed, it can be equated to an equivalent mass flow rate and should be imposed with this attribute instead of the one immediately above (see above). The unit of "scfm(air)", standardized cubic feet per minute of air, is directly available in the units menu. Therefore, no conversion is necessary to impose a standardized volumetric flow rate of air that has been corrected to the standard of 298 K and 1.01325 bar.

☉ Velocity

Velocity to be imposed at the inlet of the attached flow component or the name of a dependency reference object.

One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a velocity contour in the boundary object.

☉ Mass Flux

Mass Flux to be imposed at the inlet of the attached flow component or the name of a dependency reference object. The area which is used to calculate the total mass flow rate will be taken from the respective 'FlowOpening' to which the boundary condition is applied.





Temperature

Temperature of the fluid or the name of a dependency reference object.

One can also point to a 'CFDBoundary' or 'RLTDependenceMapXYZ' reference object to impose a temperature contour in the boundary object.

☒ Temperature Shift

Option to add a temperature offset, which can be specified as constant, time-varying, or a RLT dependence. This option is particularly useful if a variable temperature shift is needed and 'CFDBoundary' is used to specify temperature.

Composition

Name of a 'Fluid*' reference object that defines the composition of the fluid.

Initialize Flow Rate for Adjacent Connection?

When activated, this option will propagate the imposed flow rate to downstream connections. The solver will attempt to initialize the entire flow circuit to match the specified flow rate but any flow splits in the model will require that only one of the orifice connections have an undefined "Initial Mass Flow Rate" so that the proper mass flow rate can be determined by looking at the "Initial Mass Flow Rate" in each of the other connections.

Humidity

Relative Humidity (Added to specified fresh air Composition)

Relative humidity of Humidity Species (specified below) in the fresh air (i.e., excluding any burned gases and fuel in the mixture) evaluated at the initial environment temperature and an assumed 1 bar pressure. A value of "ign" results in no humidity correction to the composition.

- If dry air without any burned species is specified in the Composition attribute of the Main folder, then the Humidity Species corresponding to the entered humidity is added to the composition.
- If unburned H₂O is already specified in the Composition attribute, then only "ign" is allowed.
- If a mixture of dry air and burned species are specified in the Composition attribute, then Humidity Species will be added to produce the specified humidity for only the unburned species (i.e., dry air). This approach enables a simple way to impose the relative humidity of a mixture of fresh wet air and EGR.
- For liquid circuits, this attribute should always be set to "ign"

Recall that at a given temperature the maximum possible relative humidity is the ratio of pressure and the saturation pressure or 1,

$$\phi_{\max} = \min\left(P/P_{\text{sat}}|_T, 1\right)$$

A relative humidity greater than this maximum is not allowed. Note that a relative humidity specification should be avoided when the fluid temperature is near critical ($T_{\text{crit,H}_2\text{O}} = 647\text{K}$) because the humidity content becomes extremely sensitive to the relative humidity value (and undefined when supercritical). For these cases, H₂O can be added





	manually using a 'FluidMixture' reference object instead.
Humidity Species	'FluidGas', 'FluidNASA-LiqGas', or 'FluidPreDefined' reference object for H ₂ O vapor. Non-H ₂ O humidity species are not supported at this time. Parameters are allowed, and "ign" is permitted <u>if and only if</u> Relative Humidity is set to "ign".
☉ Apply Humidity to Initial Conditions	When choosing this option, 1 bar pressure and the temperature defined in the Main Folder are taken as the underlying reference conditions for the above specified humidity value.
☉ Reference Pressure	The entered pressure is taken as the reference pressure for the above entered relative humidity. In case the entered value and the pressure defined in the Main Folder differ, the relative humidity is re-calculated to match the imposed fluid pressure. This implies that the absolute humidity is held constant after it is calculated using the reference pressure.
(☉) Reference Temperature	The entered temperature is taken as the reference temperature for the above entered relative humidity. In case the entered value and the temperature defined in the Main Folder differ, the relative humidity is re-calculated to match the imposed fluid temperature. This implies that the absolute humidity is held constant after it is calculated using the reference temperature.





EffAreaRestriction

This template is used to restrict the area of a blockage hole that is available for flow in 'HoleVolume' or 'HoleVolumeZero'. This is typically used to model the restrictions of the grille in a fascia without having to model the individual openings.

Arrays

Effective Area Ratio

The fraction of the hole area that is open for flow. The value entered will be used to reduce the flow area of the hole opening. A value of "def" is a ratio of 1.

The value entered will be used to calculate an effective diameter from the area of the hole:

$$Eff. Dia. = \sqrt{\frac{4}{\pi} (Area_{Hole}) (Ratio_{Eff. Area})}$$

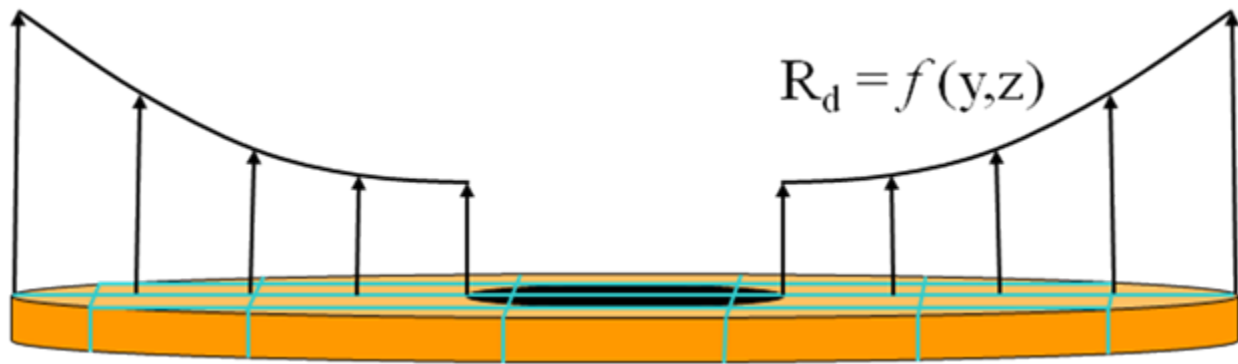




XYTableSimple

This template is a "dependency reference template", only available in COOL3D, that may be called to specify the distribution of the air flow, **Y**, through a fan as a function of the radial position, **X**. The multiplier that is applied is a surface integral of the following form:

$$\frac{\iint R_d dydz}{\iint_{total_fan} R_d dydz}, \text{ where } R_d = \text{Normalized radial distribution function}$$



Arrays

X Data

Array of the radial position of the fan measured from the edge of the hub to the outer blade. This data will be normalized to a dimensionless number from 0 to 1.

Y Data

Array corresponding to the **X Data** that defines the distribution of the air flow. This data will be normalized to a dimensionless number from 0 to 1.



Index

3

3 Points Cutting Plane, 10

A

ACIS, 48, 49, 103
 Actuator, 14, 105
 Add Arc, 26
 Add Point/Line, 25
 Ambients, 127, 132
 Area, 26
 Area Resistance, 78
 Area Restriction, 140
 Arrange Windows, 8
 Assembly Rotation, 14, 17
 Axis
 Hide, 7
 Show, 7

B

Black diamond, 12
 Blockage, 9, 69
 Boundary Conditions
 COOLEndEnv, 127
 COOLEndEnvRam, 132
 COOLEndFlowInlet, 137
 Box Selection, 5
 Build Menu, 8

C

Cancel Operation, 5
 Case Setup, 6, 18
 Center Canvas, 26
 Center Polygon, 27
 Clear All, 26
 Clip, 11
 Close All, 4
 Close Model, 3, 4
 Compare Files, 13
 Component Menu, 14
 CONVERGE Lits, 13
 Conversion
 Remove data, 12
 Convert Menu, 9
 Convert Shape, 12, 20
 Convert Shape Wizard

Blockage Conversion, 23
 Flow Space Conversion, 21
 Convert to Mesh Shape, 12
 COOLBlockage, 9, 69
 COOLDuct, 9, 71
 COOLEndEnv, 127
 COOLEndEnvRam, 132
 COOLEndFlowInlet, 137
 COOLFan, 9, 73
 COOLFlowResistance, 9, 78
 COOLFlowSpace, 8, 80
 COOLFlowSpaceSimple, 8, 83
 COOLHeatAddition, 8, 85
 COOLHeatExchanger, 8, 89
 COOLSolidBlockage, 97
 COOLSolidFlowSpace, 98
 Copy, 5
 Cross Section
 BiRadial, 9, 120
 Circle, 9, 121
 Custom, 9, 122
 Ellipse, 9, 123
 Rectangle, 9, 124
 Round Rectangle, 9, 125
 Cross Section Editor, 25
 Cross Section Ruler, 27
 Cross Section Unit, 27
 CSBiRadial, 120
 CSCircle, 121
 CSCustom, 122
 CSEllipse, 123
 CSRect, 124
 CSRoundRect, 125
 Cut, 5
 Cutting Plane, 28
 3 Points, 10
 Local, 10
 Restore, 11
 Snap to Feature, 10

D

De-convert component, 12
 Delete, 10, 26
 Delete Unused
 Objects, 13
 Objects and Template, 13
 Templates, 13
 Discretization, 6



Discretization Plane, 14, 101
 Drill Mode, 5
 Duct, 9, 71

E

Edit, 5
 EffAreaRestriction, 140
 Environments, 127, 132, 137
 Excel Spreadsheets, 14
 Exit, 3
 Export
 ACIS, 4, 36
 GTM, 6, 29
 GTSUB, 6
 Image, 4, 34
 STL, 4, 35
 Export GT Model, 6

F

Fan, 9, 73
 File Menu, 3
 File>Options, 37, 39, 40, 41, 42, 43, 44, 45
 Filter Meshes, 47
 Filter shapes, 12
 Find Template, 5
 Find Value, 5
 Fit to Screen, 26
 Flaps, 14, 110, 113
 Fan Shroud, 14, 110
 Flip
 Horizontal, 27
 Vertical, 27
 Flow Boundary Condition, 14
 Flow Opening, 14, 107
 Flow Resistance, 9, 78
 Flow Space, 8, 80
 Flow Space Simple, 8, 83
 Fluid Boundary, 107

G

GEM3D, 13
 GEMMeshShape, 102
 GEMSolidShape, 103
 Global Axis, 7
 GT-ISE, 13
 GT-POST, 13

H

Heat Addition, 8, 85
 Heat Exchanger, 8, 89
 Hole, 14
 Object, 14, 112
 Volume, 14, 111
 Zero Volume, 14, 113
 HoleDuct, 110
 HoleVolume, 111
 HoleVolumeObject, 112
 HoleVolumeZero, 113
 Home Menu, 3
 Home View, 6
 Horizontal Symmetry, 27

I

IGES, 48, 51
 Implicit Object Links
 Break All, 13
 Refresh, 13
 Import 3D, 4, 48
 Import Excel Objects, 13
 Import Shell, 48, 49, 51, 53, 55, 56, 59

L

Local Cutting Plane, 10, 61

M

Mark
 Face, 10
 Mark, 9
 Redo, 10
 Surface, 10
 Triangle, 10
 Undo, 10
 Merge meshes, 11
 Mesh
 Filter, 12
 Select all, 12
 Model Sectioning, 6
 Model View Layout, 8, 62
 Move Group, 25

N

New Model, 3
 Notes, 6



O

Open Model, 3, 4
Options, 3
Orientation, 6

P

Paint
 Face, 10
 Paint, 9
 Redo, 10
 Surface, 10
 Triangle, 10
 Undo, 10
Parameters, 6, 13, 18
Parasolid, 48, 53, 103
Paste, 5
Patch mesh ports, 12
Perspective View, 7
Polygon Vertices, 26, 63
Pressure Drop Plane, 9, 78

R

Redo, 5, 25
Reference Objects, 13
Refresh, 4
Reload, 4
Remove deconversion data, 12
Render Mode, 7
Restore cutting plane, 11
Rotate, 6, 26

S

Save As, 3, 4
Save Model, 3, 4
Select all shapes, 12
Selection
 Box, 5
 General, 5
Sensor, 14
SensorConn, 114
Separate by Curves, 11
Set Anchor Point, 27
Set Rotation Point, 8, 64

Snap to Feature Cutting Plane, 10
Snap to Grid, 25
Solid
 Filter, 12
 Select all, 12
Solid Model, 103
SPACECLAIM, 13
STEP, 103
STL, 48, 55, 56, 59
Stop Operation, 5

T

Table Edit View, 8
Template Library, 5
Template Library, 4
Tile, 8
Toggle Grid, 25
Tools Menu, 13
Translation, 14, 65

U

Undo, 4, 25
Unselect All, 5

V

Validate Drawing, 26
Vehicle
 Ram Air Velocity, 132
Vertical Symmetry, 27
View Menu, 6
View Model Sectioning, 66
VTDESIGN, 13

X

XYTableSimple, 141

Z

Zoom, 7
Zoom Default, 26
Zoom In, 26
Zoom Out, 26

