



Altair Engineering Contact Information

Web site www.altair.com

FTP site Address: ftp.altair.com or ftp2.altair.com or <http://ftp.altair.com/ftp>
Login: ftp
Password: <your e-mail address>

Location	Telephone	e-mail
North America	248.614.2425	hwsupport@altair.com
China	86.21.6117.1666	support@altair.com.cn
France	33.1.4133.0992	francesupport@altair.com
Germany	49.7031.6208.22	hwsupport@altair.de
India	91.80.6629.4500 1800.425.0234 (toll free)	support@india.altair.com
Italy	39.800.905.595	support@altairengineering.it
Japan	81.3.5396.2881	support@altairjp.co.jp
Korea	82.31.716.4321	support@altair.co.kr
Scandinavia	46.46.286.2052	support@altair.se
United Kingdom	44.1926.468.600	support@uk.altair.com
Brazil	55.11.3884.0414	br_support@altair.com.br
Australia	64.9.413.7981	anzsupport@altair.com
New Zealand	64.9.413.7981	anzsupport@altair.com

The following countries have distributors for Altair Engineering:

Asia Pacific: Indonesia, Malaysia, Singapore, Taiwan, Thailand

Europe: Czech Republic, Hungary, Poland, Romania, Spain, Turkey.

©2009 Altair Engineering, Inc. All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated to another language without the written permission of Altair Engineering, Inc. To obtain this permission, write to the attention Altair Engineering legal department at: 1820 E. Big Beaver, Troy, Michigan, USA, or call +1-248-614-2400.

Trademark and Registered Trademark Acknowledgments

Listed below are Altair® HyperWorks® applications. Copyright® Altair Engineering Inc., All Rights Reserved for:

HyperMesh® 1990-2009; HyperView® 1999-2009; OptiStruct® 1996-2009; RADIOSS® 1986-2009; HyperCrash™ 2001-2009; HyperStudy® 1999-2009; HyperGraph® 1995-2009; MotionView® 1993-2009; MotionSolve® 2002-2009; TextView™ 1996-2009; MediaView™ 1999-2009; HyperForm® 1998-2009; HyperXtrude® 1999-2009; HyperView Player® 2001-2009; Process Manager™ 2003-2009; Data Manager™ 2005-2009; Assembler™ 2005-2009; FEModel™ 2004-2009; BatchMesher™ 2003-2009; Templex™ 1990-2009; Manufacturing Solutions™ 2005-2009; HyperDieDynamics™ 2007-2009; HyperMath™ 2007-2009; ScriptView™ 2007-2009.

In addition to HyperWorks® trademarks noted above, GridWorks™, PBS™ Gridworks®, PBS™ Professional®, PBS™ and Portable Batch System® are trademarks of ALTAIR ENGINEERING INC., as is patent # 6,859,792. All are protected under U.S. and international laws and treaties. All other marks are the property of their respective owners.

Table of Contents

HyperMesh Introduction Pre-processing for Finite Element Analysis

Day 1

Chapter 1: Basic Interaction with HyperMesh	1
Section 1: Getting Started with HyperMesh	1
Section 2: Opening and Saving Files	5
Section 3: Working with Panels	13
Section 4: Organizing a Model.....	25
Section 5: Controlling the Display	39
Chapter 2: Geometry Clean-up	53
Section 1: Importing and Repairing CAD	53
Section 2: Generating a Midsurface.....	67
Section 3: Simplifying Geometry.....	73
Section 4: Refining Topology to Achieve a Quality Mesh	81
Chapter 3: Shell Meshing	95
Section 1: AutoMeshing.....	95
Section 2: Checking and Editing Mesh	111
Section 3: Batch Meshing	127

Table of Contents

HyperMesh Introduction Pre-processing for Finite Element Analysis

Day 2

Chapter 4: Solids and Hexas	137
Section 1 : Creating and Editing Solid Geometry/Generating Solid Mesh	137
Chapter 5: Tetra Meshing.....	163
Chapter 6: Analysis Setup	175
Section 1: Setting up Loading Conditions	175
Section 2: Formatting Models for Analysis	183
Chapter 7: Connectors	195
Exercise 1: Connectors Introduction using Spot Connectors	199
Exercise 2: Creating Area Connectors.....	217
Exercise 3: Creating Bolt Connectors	222

Chapter 1

Basic Interaction with HyperMesh

Section 1: Getting Started with HyperMesh

In this section, you will explore the basic layout of the HyperMesh 9.0 user interface.

Overview of Finite Element Analysis

Finite Element Analysis was first developed over 60 years ago as a method to accurately predict the reaction of complex parts to various inputs. Prior to the development of FEA the only way to validate a design or test a theory was to physically test a part. This was and still is both time consuming and expensive. While FEA will never replace the final physical testing and validation of a design, it can drastically reduce the time and money spent on intermediate stages and concepts.

FEA in its infancy was limited to large scale computing platforms but the development of powerful personal computers combined with intuitive software packages such as HyperWorks have brought FEA to the engineers desktop and has broadened its use and accuracy many fold.

Finite Element Analysis is now a vital and irreplaceable tool in many industries such as Automotive, Aerospace, Defense, Consumer Products, Medical, Oil and Gas, Architecture and many others.

FEA is performed in three stages, Pre-Processing, Solving and Post Processing and those are outlined below.

Step 1: Pre-Processing

Pre-Processing is the act of preparing (meshing) a model for analysis. Complex geometry is broken down into simple shapes (elements) in the act of meshing. This allows the solver in the next step to predict the action of these elements and analyze the reaction of a complex part to external forces and interactions. The part is meshed and then definitions for the type and thickness of the material(s) are added then forces and constraints are applied. The model is then prepared for the analysis with information the solver will need to perform its calculations. The model is then written in a format that the solver can understand and is sent to the solver for processing.

Step 2: Solving

Solving is performed by any of the many commercially available software written to perform Finite Element Analysis. Some of these include popular packages such as Radioss, Nastran, LS-Dyna, Abaqus, Ansys and a few others. The solver takes the information provided in the file (input deck) created in HyperMesh in step one and calculates the parts reactions to the inputs defined. Common outputs are Displacement, Stress, Strain and Acceleration. These results are stored in a file that then can be read in HyperView in the Post-Processing stage.

Step 3: Post-Processing

Post-Processing is where the results of the solver solution can be reviewed. HyperView can provide presentation quality color contoured plots and animations highlighting any of the requested results. Information can be queried, displaced and even graphed in numerous windows allowing for customization geared toward the desired audience.

HyperMesh Introduction

Starting HyperMesh

- On PC
 - From the Start Menu, select All Programs>Altair HyperWorks(version)>HyperMesh.
 - Or
 - User can create a Windows Shortcut by Right Clicking on the above program and selecting Create Shortcut.
- On UNIX
 - At the prompt, type <install directoy>/scripts/hm.
 - Set up an alias.

The Start Directory

The “Start Directory” or “Working Directory” is where HyperMesh, by default, looks for and saves the following files;

- Configuration files (*hmmenu.set*, *hmsettings.tcl*, *hm.mac*, *etc.*)
- History File (*command.cmf*)
- HyperMesh Model Files, FE Data and Geometry Files. (User can browse to different directories for opening and saving.)

The Start Directory can be defined as follows:

On PC

1. Right Click Shortcut link
2. Select Properties
3. Enter the desired directory in the Start In field or leave blank to default to directory where the shortcut is located

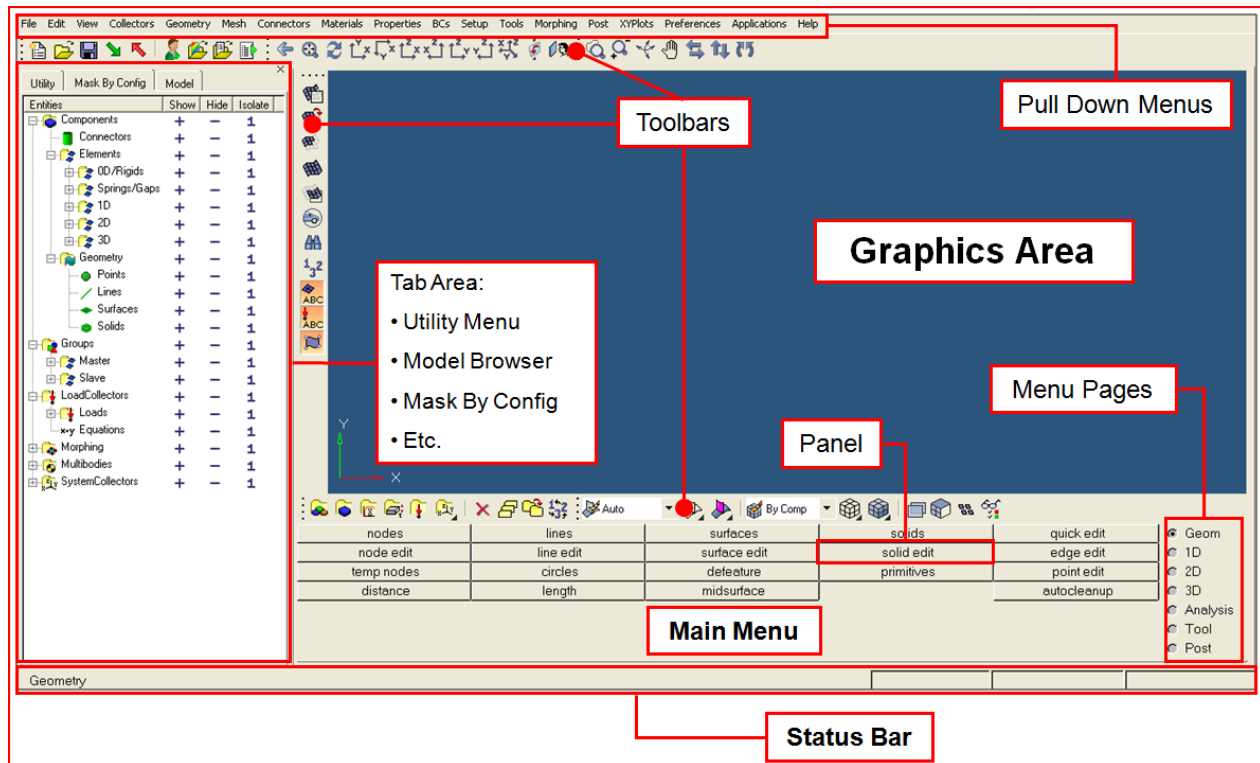
On UNIX

- The Start In directory will default to the Directory where the program is executed from. If the configuration files are not found in that directory, then HyperMesh will look in the users “home” directory.

Online Help

HyperMesh offers comprehensive documentation in the online help. The Help can be accessed through the Pull Down menu or the use of the “h” key on your keyboard. If the user accesses help through the use of the “h” key the help documentation is “intelligent”, opening in the section representing the panel that the user is actively in. Help also contains detailed tutorials on many advanced HyperMesh functions.

HyperMesh Graphical User Interface



- **Graphics area** – displays the model
- **Toolbar** – gives access to commonly used tools via icons
- **Pull Down Menu** – places functionality into groups, accessible via pull downs
- **Menu Pages** – divides the main menu into groups based on function
- **Main Menu** – contains “panels” grouped in columns
- **Panels** – menu items / functions for interacting with HyperMesh
- **Sub-panels** – divides panel into similar tasks related to panel's main function
- **Command Window** – lets the user type in and execute tcl commands
 - Available through the **View** drop down menu (turned off by default)
- **Tab Area** – contains the following tabs:
 - Solver, Model, Utility, Include, Import, Export, Connector, Entity State, etc.
- **Status Bar** – shows status of operations being performed
 - Indicates the “current” Include file, Component Collector, and Load Collector

Section 2: Opening and Saving Files

Bringing data files into HyperMesh and saving them are frequent operations every user should understand. This section will help you become proficient with the various ways this can be done in HyperMesh. The remaining exercises in this course will assume you know how to open and save files in HyperMesh.

In this section, you will learn how to:

- Open a HyperMesh file
- Import a file into a current HyperMesh session
- Save the HyperMesh session as a HyperMesh model file
- Export all the geometry to an IGES file
- Export all the mesh data to a RADIOSS input file
- Delete all data from the current HyperMesh session
- Import an IGES file
- Import a RADIOSS file to the current HyperMesh session

File Operations



New .hm File – Creates a new session in the current window



Open .hm File – Loads a HyperMesh model into the current window replacing the current model



Save .hm file – Saves the current model, opens browser window.



Import – Opens the Import Tab allowing the import of;



.hm Models



FE Models



Geometry Files



Connectors

into the current model.



Export – Opens the Export Tab allowing the export of:



FE Models



Geometry Files



.h3d Files



Connectors

Opens a browser window.



Load User Profile – Opens the User Profile Window.



Load Results File – Loads a result file for post processing within HyperMesh.



Open Current Command File – Opens a window displaying the current command.cmf file. Can be used to learn TCL/Tk commands and create macros.



Run TCL Script/Command File –



Left Mouse Click opens a browser to load and run a TCL file.




Right Mouse Click opens a browser to load and execute a command.cmf file.

Exercise: Opening and Saving Files

This exercise uses the following model files: bumper_cen_mid1.hm, bumper_mid.hm, bumper_end.iges, and bumper_end_rgd.fem. Each model file contains various sections, That will be assembled into the full bumper model.

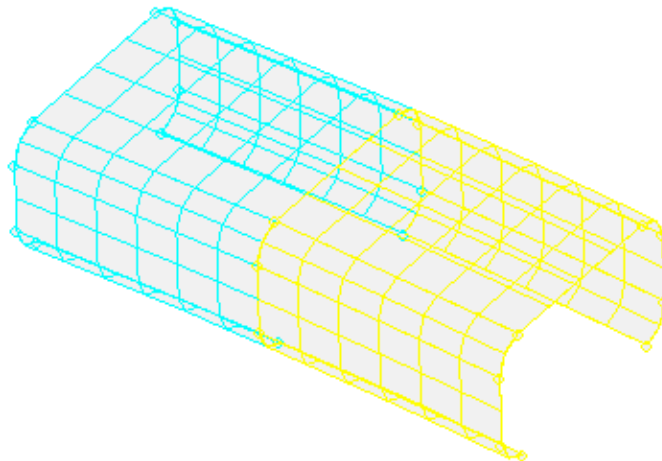
Step 1: Open the HyperMesh model file, bumper_cen_mid1.hm.

1. Access the **Open File...** dialog in one of the following ways:

- From the **Pull-down** menu, choose **File**, then **Open**
- From the **Standard toolbar**, click **Open .HM File** 

2. Open the model file, bumper_cen_mid1.hm.


The model file, bumper_cen_mid1.hm, is now loaded. This file contains mesh and geometry data.





HyperMesh model file, bumper_cen_mid1.hm, opened in HyperMesh

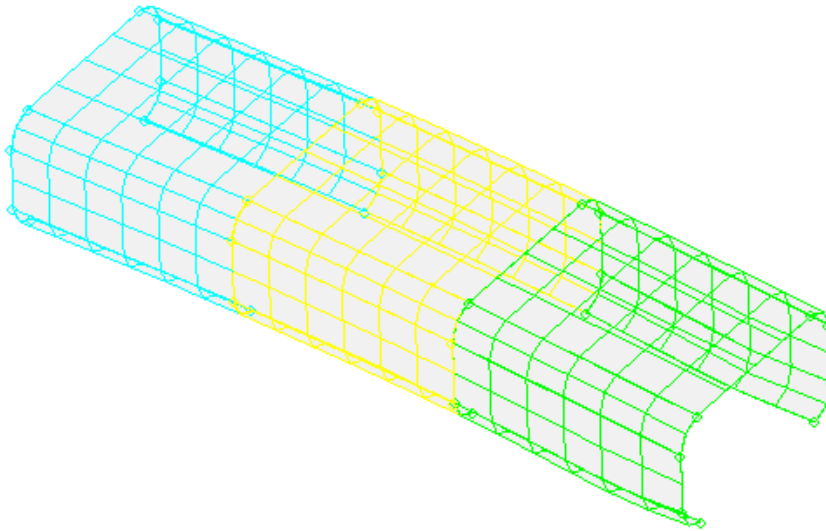
Step 2: Import the HyperMesh model file, bumper_mid.hm, into the current HyperMesh session.

1. Access the **Import** tab in one of the following ways:

- From the **Pull-down** menu, choose **File**, then **Import**.
- From the **Standard toolbar**, click **Import** 



2. From the **Import** tab in the **tab** area, choose the **HM Model icon**  from the selection.
3. Under **File selection**, click the **Open**  and browse to select the file, bumper_mid.hm.
4. Click **Import**.

The file, bumper_mid.hm, is now imported into the session.

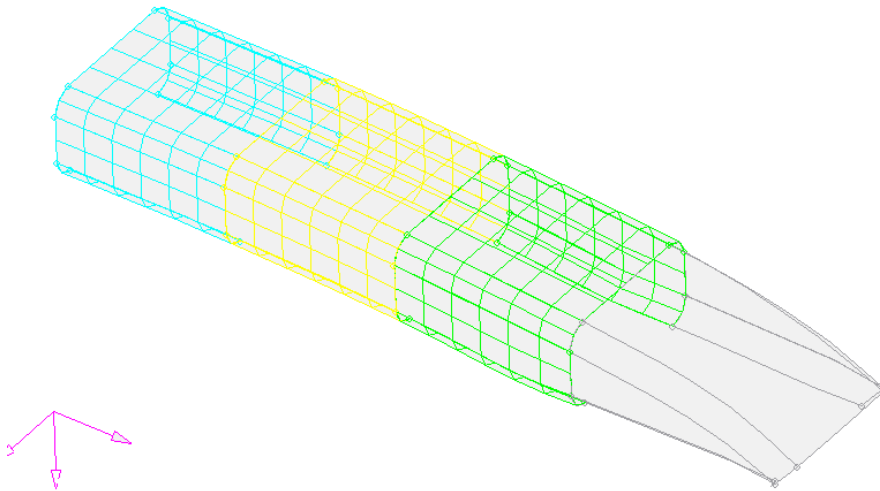


HyperMesh model file, bumper_mid.hm, imported on top of existing data in the HyperMesh session

Step 3: Import the IGES geometry file, bumper_end.iges, into the current HyperMesh session.


1. From the **Import** tab in the **tab** area, choose the **Geometry icon** .
2. Under **File selection**, choose **File type: Iges** from the **pull-down** menu.
3. Under **File selection**, click the **Open** icon  and browse to select the file, bumper_end.iges.
4. Click **Import**.

Geometry data is added to the model.

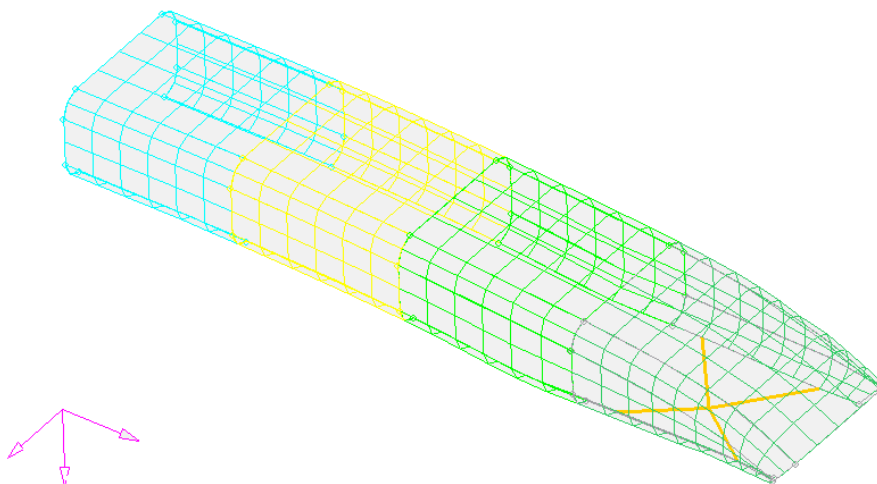


IGES geometry file, bumper_end.igs, imported into the session

Step 4: Import the RADIOSS Bulk Data input file, bumper_end_rgd.fem, into the current HyperMesh session.

1. From the **Import** tab in the **tab** area, choose the **FE Model icon** .
2. Under **File selection**, select **File type: RADIOSS (Bulk Data)** from the pull-down menu.
3. Under **File selection**, click the **Open** icon in **File:** and browse to bumper_end_rgd.fem.
4. Click **Import**.

This RADIOSS input file contains mesh for the bumper's end portion. The mesh is added to the existing data in the current HyperMesh session and will be located in the same area as the geometry representing the bumper's end.




RADIOSS input file, bumper_end_rgd.fem, imported on top of data in the current HyperMesh session


Step 5: Save the HyperMesh session as a HyperMesh model file called `practice.hm`.

1. From the **File** pull-down menu, select **Save As...**
2. Enter the name, `practice.hm`.
3. Click **Save**.

The data currently loaded in HyperMesh is now saved in a HyperMesh binary data file of the name you entered.

Step 6: Export the model's geometry data to an IGES file called `practice.igs`.

1. Access the **Export** tab in one of the following ways:
 - From the **Pull-down** menu, choose **File**, then **Export**.
 - From the **Standard toolbar**, click **Export** .

2. In the **Export** tab, choose the **Geometry icon** .

3. Under **File selection**, **File type:** should indicate **Iges**.
4. Under **File selection**, click the **Open** icon in the **File** field, browse to the desired destination folder, and enter `practice.igs`.
5. Click **Save**.

All of the geometry loaded in HyperMesh (points, lines, surfaces) is now saved in an `.iges` file with the name you entered.

6. Click **Apply** to export the file.


Step 7: Export the model's mesh data to a RADIOSS Bulk Data input file called `practice.fem`.

1. In the **Export** tab, choose **Export type: FE Model** from the pull-down menu.
2. Under **File selection**, choose **File type: RADIOSS (Bulk Data)** from the pull-down menu.
3. Under **File selection**, click the **Open** icon in the **File** field, browse to the desired destination folder, and enter `practice.fem`.
4. Click **save**.

All of the finite element data loaded in HyperMesh (nodes, elements, loads, etc.) is now saved as an `.fem` file with the name you entered.

5. Click **Export**.

Step 8 (Optional): Delete all data from the current HyperMesh session.

1. Access the delete **New HyperMesh Model** function in one of the following ways:
 - From the **Pull-down** menu, click **File**, then **New...**
 - From the **Standard toolbar**, click **New .hm File** 
2. Answer **Yes** to the pop-up question "Do you wish to delete the current model? (y/n)".

Step 9 (Optional): Import the IGES geometry file you created, `practice.igs`.

Refer to Step 3 for detailed instructions.

Step 10 (Optional): Import the RADIOSS Bulk Data input file you created, `practice.fem`, into the current HyperMesh session.

Import `practice.fem` into the current session. The data in the file will be added to the existing data in the current HyperMesh session. Refer to Step 4 for detailed instructions.

With the completion of Steps 8, 9, and 10, your current HyperMesh session should contain all of the geometry and mesh data that existed in the HyperMesh session that was saved to a HyperMesh file in Step 5.

Step 11 (Optional): Save your work.

Section 3: Working with Panels

Much of the functionality in HyperMesh is centered around the use of panels. While there are often many ways to get to a function within HyperMesh, most often the actions lead the user to the panel area to select entities, enter values and execute functions. The panel area is split into seven pages and on each page are panels that allow the user to utilize all of the functionality in HyperMesh. Even if the user accesses a function through the use of the menu bar or the toolbars, much of the information will be entered in the panel area. While this manual cannot explain the functionality of every panel, much of the panel functionality is common amongst all of the panels and thus learning one panel will assist the user in the use of all panels.

This section introduces you to common panel attributes and controls as it guides you through translating nodes and elements using the ***translate*** panel and measuring distances between nodes using the ***distance*** panel.

In this section, you will learn how to:

- Use the entity selector and the extended entity selection menu to select and unselect nodes and elements from the graphics area.
- Use the direction selector to define vectors along which to translate nodes and elements.
- Switch between different entities to select, and methods to define vectors.
- Toggle between two options.
- Enter, copy and paste, and calculate numbers.
- Use the rapid menu functionality to execute commands with the mouse buttons rather than clicking buttons.
- Interrupt, but not exit, a panel to go to another panel using the keyboard function keys.

Panel Layout

In HyperMesh, panels have three general layouts. The Basic Panel, Panel with Sub-Panels and Panels with Sub Panels organized in Columns. Their look and functionality will be described below.

• The Basic Panel

Translate Panel

• Panel with Sub-Panels

Project/to plane Panel

• Panel with Sub-Panel and Columns

Surface Edit/trim with surfs/planes

Generally panels are used in a Left to Right manner and those with columns are used in a left to right and top to bottom manner using the following steps:

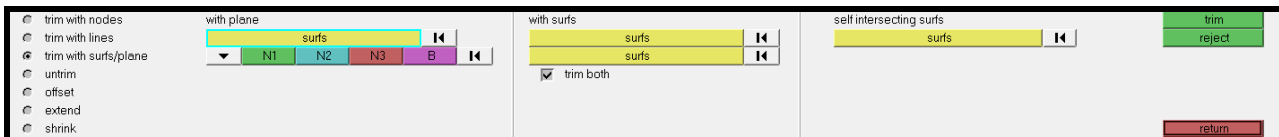
Step 1: “What to Do”

This step only applies to panels with Sub Panels. The user picks the functionality within the panel that is desired by picking the appropriate Sub Panel radio button. The example below is from the Project panel and the “to plane” sub functionality is chosen.

Step 2: “Method to Use”

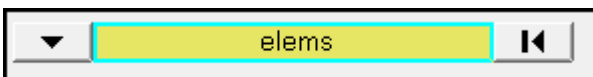
This step only applies to Panels with Sub Panels that are organized in columns. Often, Sub Panels are organized into different columns when there are more than 7 Sub Panel options. The column organization groups like functionality together in instances where the entire

panel is not needed for information entry. In this case the user picks the Sub Panel in Step 1 and then chooses the method they wish to use within that sub panel and follows the column top to bottom. The example below shows the Surface Edit panel with the trim with surfs/plane sub functionality chosen. You can see the three columns providing access to either the “with plane”, “with surfs” or “self intersecting surfs” options.



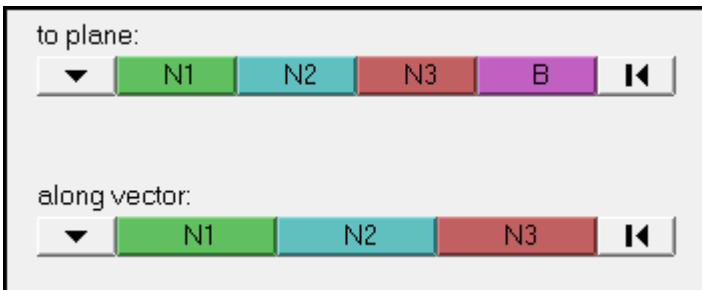
Step 3: “What to do it to”

In this step the user will select the entities they wish to perform the function on. The entity selection is shown below.



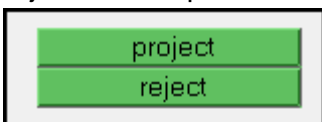
Step 4: “How to do it”

In this step the user defines parameters that dictate how the function will be performed.



Step 5: “Do the action

Clicking the green “action” button performs the desired function while the “Reject” button will reject the last performed function.



Tools within the Panels

Within the panels there are many buttons and options that will be explained below:

Switches -

These allow the choice of multiple options through a pop up menu

Toggles -

The toggle will change the function between 2 options.

Reset - 

This will reset the selection of any entities.

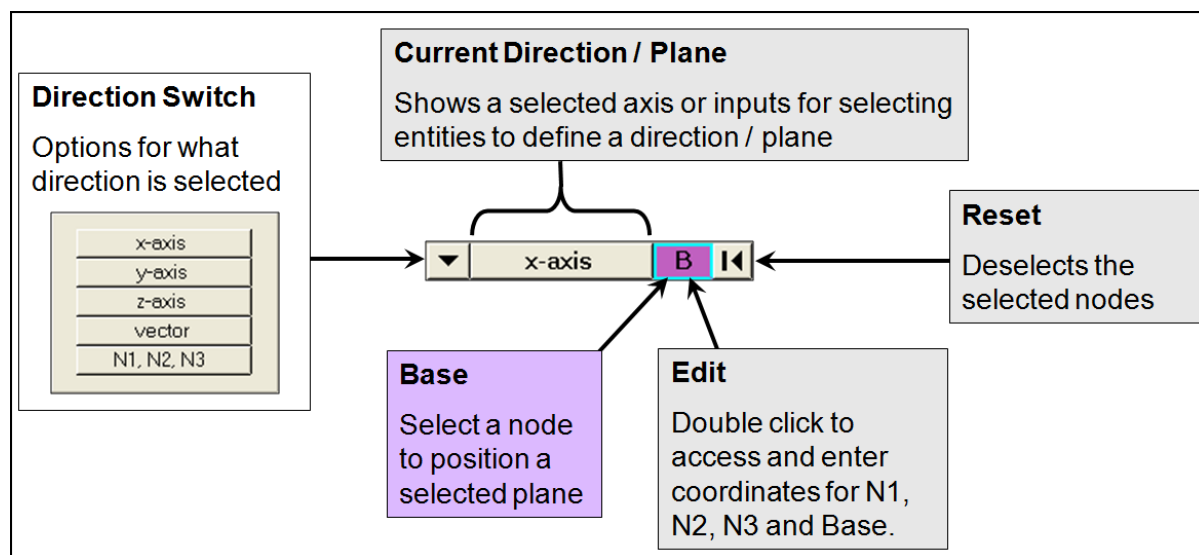
Extended Selection - 


Clicking the yellow selection button will open the extended selection window. This provides numerous tools allowing for the advanced selection of entities.

by window	on plane	by width	by geoms	by domains
displayed	retrieve	by group	by adjacent	by handles
all	save	duplicate	by attached	by block
reverse	by id	by config	by face	by path
by collector	by assems	by sets	by outputblock	by include

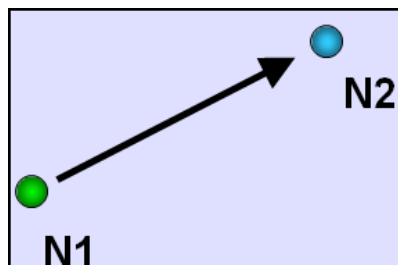
Direction/Plane Selection -

- **X, Y, Z Axis -** 

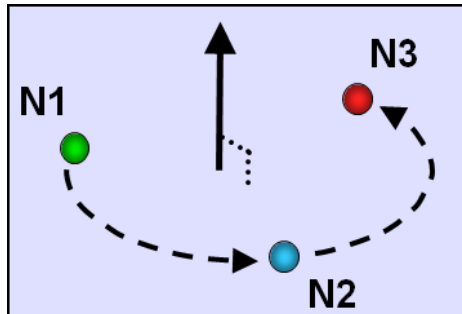


- **N1, N2 and N3 -** 

- **Select 2 Nodes (N1 & N2)** – This defines a directions from N1 to N2 where a vector type direction is required. When a plane is required the plane is defined as that which is normal to the vector N1 to N2 and its location at the B node.




- **Select 3 Nodes (N1, N2 and N3)** – This defines a plane whose normal defines a direction when a vector direction is required. Positive of the normal is defined by the “Right Hand Rule.” In the event a plane is required the plane is that which is created by the three nodes with its location at the B node.



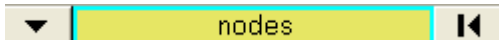
Exercise: Using the Translate Panel

Step 1: Retrieve the HyperMesh model file, bumper.hm.

1. Access the **File** panel in one of the following ways:
 - On the **File** menu, click **Open**
 - On the toolbar, click the **open** icon  and go to the **hm file** sub-panel
2. Click **retrieve...** and browse to <install directory>/tutorials/hm/bumper.hm
3. Click **Open**.

Step 2: In the *translate* panel, select nodes from the graphics area.

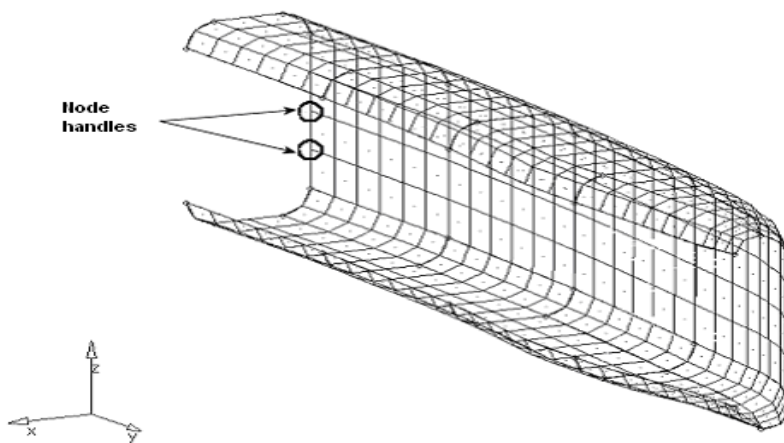
1. Access the **translate** panel in one of the two following ways:
 - From the **Pull-down menu**, select **Geometry** or **Mesh** and then choose **Translate>nodes**
 - From the **Tool** page, access the **translate** panel.
2. With the **nodes** selector active, select a few nodes from the Graphics area by left-clicking the corners of the elements.



Node selector

The cyan border around it indicates that it is active; HyperMesh is expecting nodes to be selected as the next action.


A node is positioned at each element corner. A selected node is highlighted with a small, white circle.

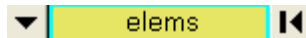


Node Handles

3. Reset the selection of nodes by clicking the **reset** icon, .

Step 3: Select and unselect elements from the graphics area.

1. Click the **entity selector switch** () and select **elems**. (Switch the entity selector to **elems**.)



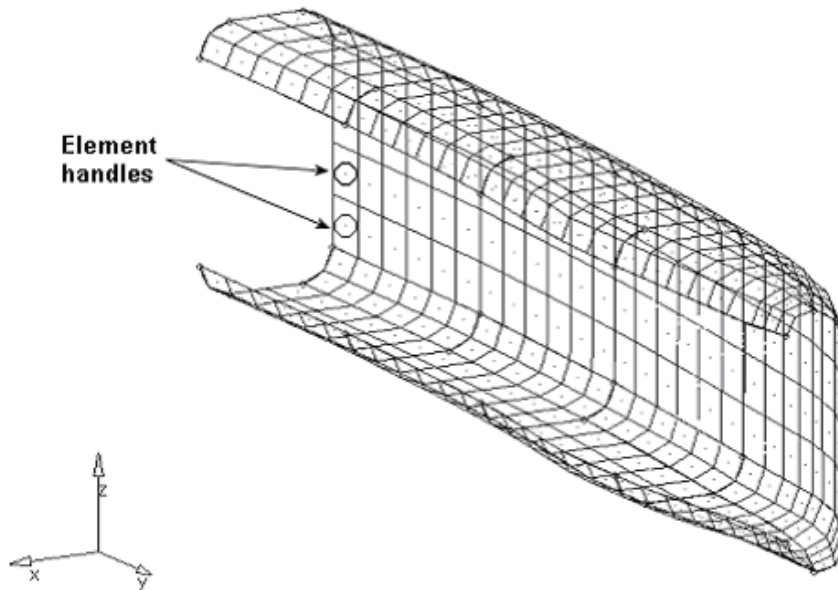
Entity selector with its switch

The menu that pops-up contains a list of entities that can be translated.

2. With the **elems** selector active, select several elements from the graphics area.

To select an element, click its element handle (the dot at the element's center).

The elements are highlighted in white when they are selected.



Element Handles

3. Unselect an element using the right mouse button.

Step 4: Select and unselect elements using the quick window selection method.

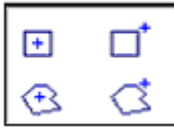
1. Verify that the **elems** selector is active.
2. Move the mouse handle into the graphics area.
3. Press and hold the **SHIFT key + left mouse button** and move the mouse to draw a rectangular window around a few elements, and then release the SHIFT key and left mouse button.

All the element handles inside the rectangular window are selected.

4. Unselect elements by pressing and holding the **SHIFT key + right mouse button** and moving the mouse to create a window around the selected elements.

- Press the SHIFT key and quick-click the left mouse button.

A pop-up window appears, which contains four icons as shown in the following image.



Quick window pop-up menu

- Select the inside polygon shape.
- Press and hold the SHIFT key + left mouse button and move the mouse around a few unselected elements, and then release the SHIFT key and mouse button.

This draws a polygon window rather than a rectangular window. All element handles inside this window are selected.

Step 5: Select and unselect elements by using the extended entity selection menu.

- Click the **elems** selector and select **reverse**. (Select **elems >> reverse**.)

The selection of elements is reversed; the elements that were selected are now unselected and the elements that were not selected are now selected.

The menu that appears contains a list of functions for selecting elements. Once you select a function from the menu, the menu disappears. If you do not want to select a function, move the mouse handle out of the menu.



by window	on plane	by width	by geoms	by domains
displayed	retrieve	by group	by adjacent	by handles
all	save	duplicate	by attached	by block
reverse	by id	by config	by face	by path
by collector	by assems	by sets	by outputblock	by include

Extended entity selection menu

- Select **elems >> by adjacent**.

Elements adjacent to the selected elements are now selected.

Step 6: Shade the elements, reset the selection, and select a few adjacent elements.

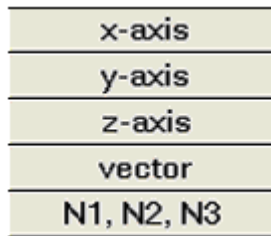
- On the **Visualization** toolbar, click the **Shaded Elements & Mesh Lines** icon, .
- The elements are displayed in shaded mode, rather than wireframe mode.
- In the **translate** panel, click the **reset** icon () to clear the elements selection.
- With the **elems** selector active, select a few elements that are adjacent to each other.

Step 7: Specify a direction vector (N1 and N2 only) along which to translate the selected elements.

Click the **direction selector switch** (,  **N1**  **N2**  **N3**  **B** ).

Direction selector along with the node selectors to define the direction vector

The menu shown in the following image appears. It contains a list of vector and plane options for defining the direction in which to translate the selected elements.



Direction selector pop-up menu

3. Select **N1**, **N2**, **N3** from the pop-up menu.

N1 now has a cyan border indicating it is the active selector.

The selected elements are displayed in gray because the **elems** entity selector is not active.

4. In the graphics area, select any node for **N1**.

The selected node is highlighted in green. The active selector advances to **N2**.

5. Select a node near **N1** for **N2**.

The selected node is highlighted in blue. The active selector advances to **N3**. Do not select a node for **N3**.

Note: Selecting the two nodes, **N1** and **N2**, defines a vector for the direction of translation. This vector goes from **N1** towards **N2**. Selecting a third node, **N3**, defines a plane. The direction of translation is the positive direction of the vector normal to the plane. The positive direction is determined by the right-hand rule.

Step 8: Specify a distance to translate the selected elements and then translate them.

1. Click the **toggle** () to change **magnitude =** to **magnitude = N2-N1**.

2. Click **translate +**.

The selected elements translate in the direction from **N1** to **N2** by **N2-N1** units.

3. Notice the thick, black border around the **translate** button. It indicates this is a rapid menu button; you can click the middle mouse button rather than click **translate +**.

4. Click the middle mouse button.

The selected elements are translated again by **N2-N1** units.

5. Click **translate -** twice.

The selected elements are translated in the negative **N1-N2** vector direction and are now in their original position.

Step 9: Measure the distance between two nodes.

1. Press the F4 function key to interrupt, but not exit, the **translate** panel and go to the **distance** panel on the **Geom** page.

The element and node you selected in the **translate** panel are currently not visible. However, they are still selected. They will be visible again when you return to the **translate** panel.

2. Verify you are in the **two nodes** sub-panel.

Notice **N1** is the active selector.

3. Select any node for **N1**.

The entity selector advances to **N2**.

4. Select a node near **N1** for **N2**.

Notice the **distance =** field value reflects the absolute distance between **N1** and **N2**.

5. Click in the **distance =** field to highlight the value.
6. Press CTRL+C to copy the value.
7. Click **return** to return to the **translate** panel.
8. Notice the elements and nodes you selected in the **translate** panel before you went to the **distance** panel are once again visible.

Step 10: Specify a distance to translate the selected elements and then translate them.

1. Toggle from **magnitude = N1-N2** to **magnitude =**.
2. Click in the **magnitude =** field to highlight its value.
3. Press CTRL+V to paste the distance = value copied from the distance panel.
4. Click **translate +**.

The selected elements translate in the direction from **N1** to **N2** by the number of units specified for **magnitude =**.

5. Click **translate -** once.

The selected elements are translated in the negative **N1-N2** vector direction and are now in their original position.

Step 11: Calculate 5.5 * 10.5 and specify the resulting value for **magnitude =**.

1. Double-click **magnitude =**.
2. Click **5 . 5** (in that order) and then click **enter**.
3. Click **10 . 5** (in that order).

4. Click **X**.


The calculated value in the calculator window is 5.775 e+01.

5. Click **exit**.

The calculator closes and 57.75 appears in the **magnitude =** field.

You can type a value in the **magnitude =** field by clicking in the field once to highlight the current value. Then type a new value.

Step 12: Specify a new vector and translate the elements again.

1. Click the **reset** icon, , for the **direction** selector. ()
2. Notice **N1** is the active selector.
3. Select three nodes for **N1**, **N2**, **N3** to define a plane.
4. Click **translate +** or press the middle mouse button.

The elements are translated 57.75 units in the positive direction normal to the defined plane.

5. Click **return** to return to the main menu.

Step 13 (Optional): Save your work.

With all of the exercise complete, you can save the model if desired.

Section 4: Organizing a Model

Organizing model data can be beneficial when creating a valid solver input file. Basic tasks used to organize model data such as placing elements and loads into groups (collectors), organizing collectors into assemblies, renaming, deleting, reordering, and renumbering are discussed in this section.

In this section, you will learn how to:

- Create geometry and organize it into components.
- Organize elements into the components.
- Rename components.
- Identify and delete empty components.
- Delete all the geometry lines.
- Reorder the components in a specific order.
- Renumber all the components, starting with ID 1 and incrementing by 1.
- Create an assembly.
- Organize the constraints.

Model organization is at the heart of a quality Finite Element Analysis. The model can be organized in a multitude of different ways as desired by the user, but below are the basics for model organization.

Collectors

The basis for model organization is the collector. HyperMesh has 10 different types of collectors;

Component – Contains Elements and Geometry

Multibody – Contains Ellipsoids, Mbjoints, Mbplanes and Sensors.

Assembly – Contains Components, Multibodies and Assemblies.

Load – Contains Loads and Constraints.

Property* – Used to define the properties assigned to elements or Component Collectors

Material* - Used to define the material values Property Collectors

System – Contains user defined systems.

Vector – Contains vectors.

Beam Section – Contains beam sections.

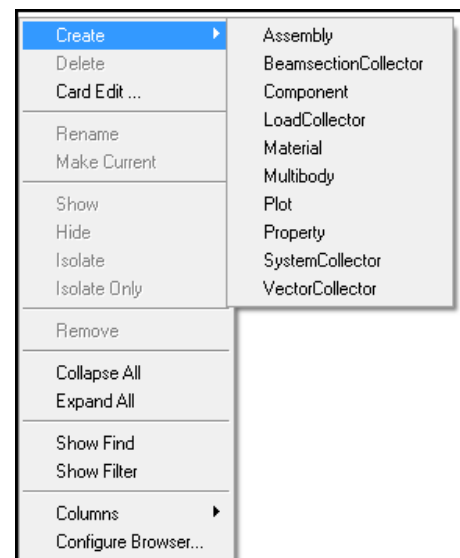
**NOTE: Property and Material collectors do not “contain” any entities and are used to define material and physical properties in the model. They are called collectors for uniformity.*

Collectors can be created in a number of ways.

- **Model Browser:**

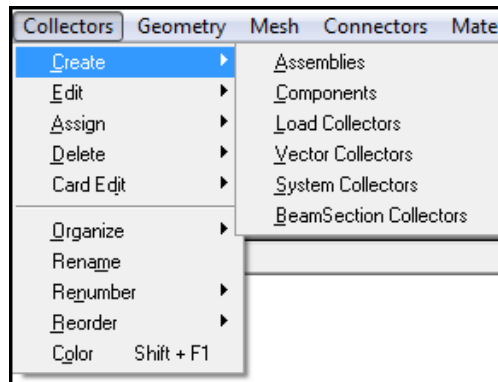
Right Clicking in the Model Browser opens a menu from which the selection of Create allows for creation of any type of collector.

Right clicking in the Model Browser will also allow you to edit, rename, change ID, change color and delete collectors as well.

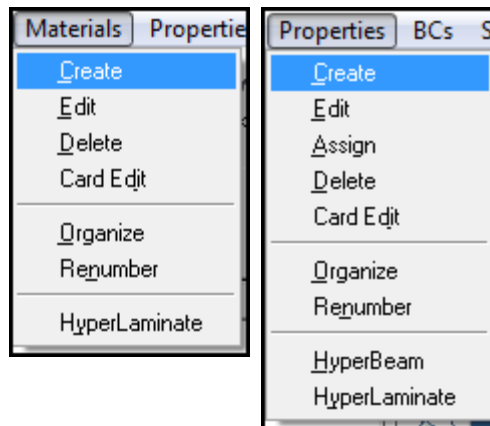


- **Pull down Menus**

Selecting the Collectors Pull-Down and then selecting Create will provide the ability to create any of the non property collectors.

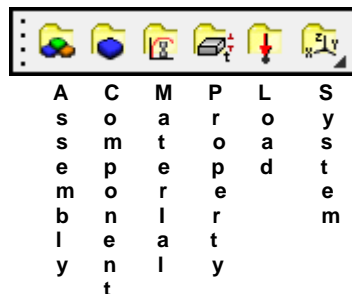


Material and Property Collectors can be created in a similar manner using the Material and Property Pull-Downs.



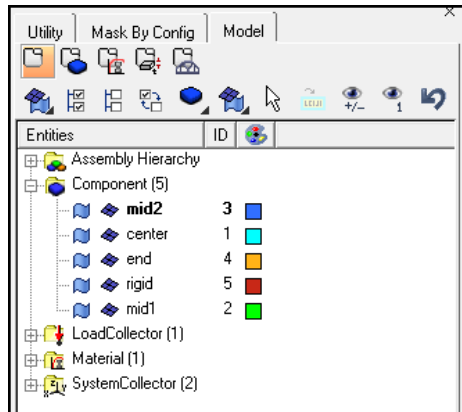
- **Icon Toolbars**

The Icons can be used to create collectors as well.

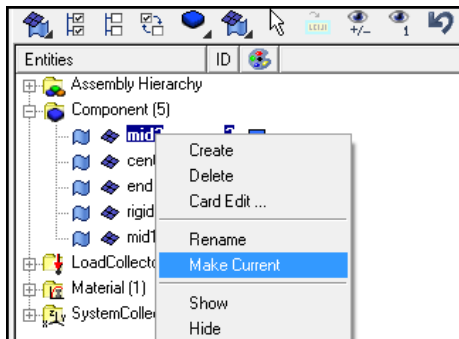


Current Collector - The Current Collector determines what collector new entities are placed in. The Current Collector can be determined in two ways.

- In the Model Browser the Current collector will be in **BOLD**



Note the **mid2** collector is in a bold font and thus is the Current collector.

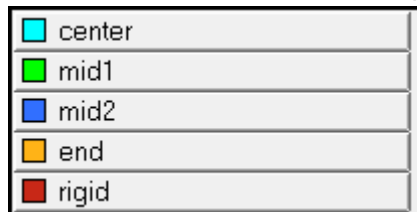


Right clicking on a collector will open a menu that will allow it to be made current.

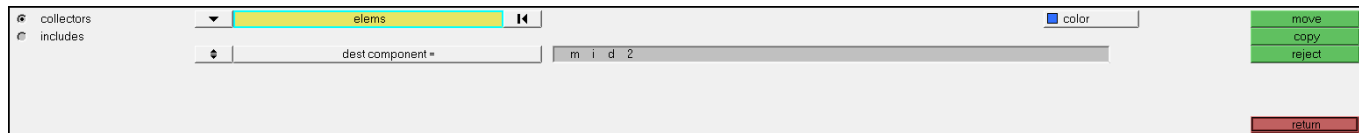
- On the message bar at the right of the screen the Current Collector is shown



This can be clicked to open a window where the Current Collector can be changed

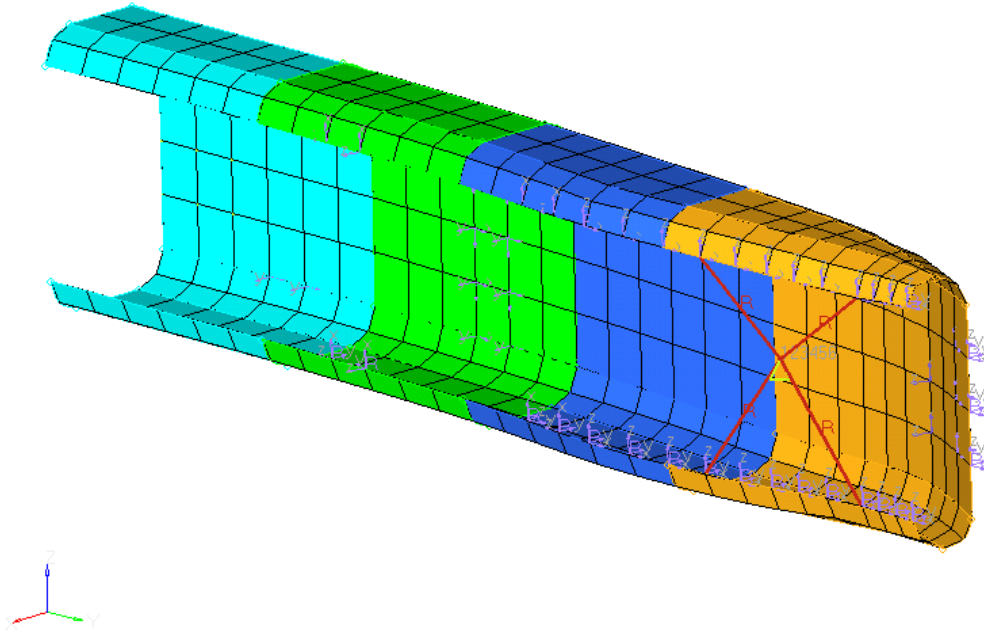


Organize – Organize is a tool that can be used to move entities to different collectors. It can be accessed on the Tool page and will open the panel shown below.




Exercise: Reorganizing a Bumper Model

Step 1: Retrieve the model file, bumper.hm.



Step 2: Create a component named *geometry* to hold the model's geometry.

1. Access the **component collector** panel in one of the following ways:
 - From the **Pull-down** menu, choose **Collectors** then **Create** and select **Components**.
 - On the **Collectors** toolbar, click **component collector** ().

2. Go to the **create** sub-panel.
3. For **comp name=**, enter *geometry*.
4. Click **color** and choose yellow.
5. Leave the **card image=** field blank.
6. Click **create** to create the component collector, *geometry*.

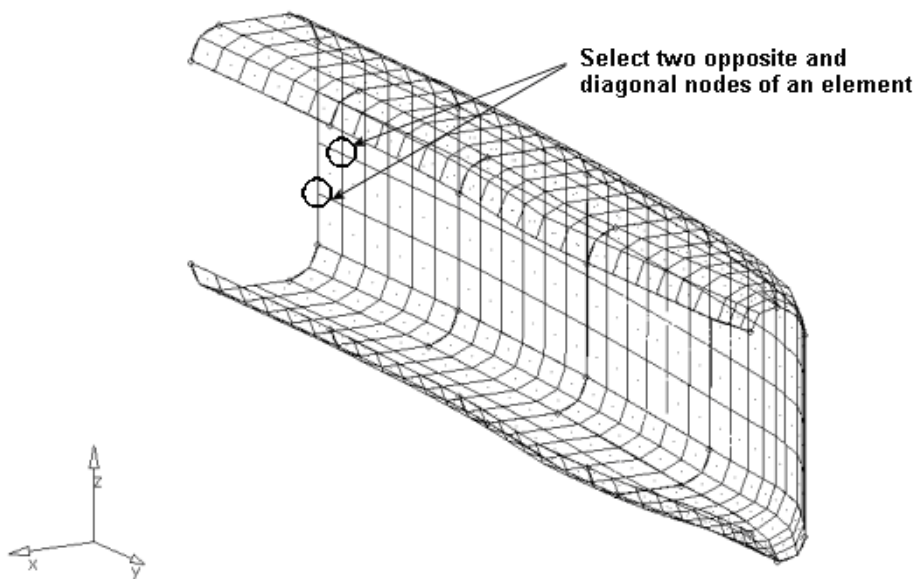
Left-click once anywhere in the HyperMesh window (except on a button) to dismiss the message in the **header** bar.

The component called **geometry** now appears in the **status** bar. It is now the current component. Any geometry or elements that are created will be organized into this component.

7. Toggle **Property Type** to **No Property**
8. Click **return** to return to the main menu.

Step 3: Create two geometry lines and organize them into different components.

1. Access the **lines** panel in one of the following ways:
 - From the **Pull-down** menu, choose **Geometry**, select **Create**, and click **Lines**.
 - From the main menu, choose the **Geom** page, and click **lines**.
2. Go to the **from nodes** sub-panel.
3. With the **node list** selector active, select two nodes, opposite and diagonal to each other, on the same element as indicated in the following image.



4. Click **create** to create the line.
5. Notice the line is yellow, the same color assigned to the component, **geometry**. This is because the line is organized into the current component, **geometry**.
6. On the **status** bar, located at the bottom of the screen, click **geometry**.
7. From the list of components in the model, click **rigid**.
8. The **status** bar now shows the current component as **rigid**.
9. With the **node list** selector active, select two nodes, opposite and diagonal to each other on another element.
10. Click **create** to create the line.
11. Notice the line is dark pink, the same color assigned to the component, **rigid**. This is because the line is organized into the current component, **rigid**.
12. Click **return** to return to the main menu.

Step 4: Move all the model's geometry surfaces into the component, **geometry**.

1. Access the **Organize** panel in one of the following ways:

- From the **Pull-down** menu, select **Geometry** or **Mesh** then choose Organize>surfaces
 - From the **Tool** page, access the **organize** panel
2. Go to the **collectors** sub-panel.
 3. Switch the **entity** selector to **surfs**.
 4. Select **surfs >> all**.
Displayed surfaces are highlighted in white indicating they are selected. All other surfaces that are not displayed are still selected because you selected **surfs >> all**.
 5. Click **dest =** and select the component, **geometry**, from the list of components in the model.
 6. Click **move** to move the selected surfaces into the component, **geometry**.

Step 5: Move all the model's shell elements (quads and trias) into the component, **center**.

You should still be in the **organize** panel.

1. Switch the **entity** selector to **elems**.
2. Select **elems >> by collector**.
A list of the model's components appears.
3. Select the components, **mid1**, **mid2**, and **end**.
Select a component by left-clicking its name, color, or check box. A component is selected when it has a check in its check box. To unselect a component, right-click it.
4. Click **select** to complete the selection of components.
5. Set **dest =** to the component, **center**.
6. Click **move** to move the elements in the selected components to the component, **center**.
All of the shell elements should now be a cyan blue, the same color assigned to the component, **center**.
7. Click **return** to return to the main menu.

Step 6: Rename the component, **center**, to shells.

1. Access the **Rename** panel in one of the following ways:
 - From the **Pull-down** menu, select **Collectors** then choose **Rename**
 - From the **Tool** page, enter the **rename** panel.
2. Go to the **individually** sub-panel.
3. Verify the entity type is set to **comps**.
4. Click **original name =** and select the component, **center**, from the list of the model's components.

5. Click once in the **new name =** field to highlight its text.
6. Type `shells`.
7. Click **rename** to rename the component.
8. Click **return** to return to the main menu.

Step 7: Identify and delete all of the empty components.

1. Access the **Delete** panel in one of the following ways:
 - From the **Pull-down** menu, select **Collectors** then choose **Delete>components**
 - From the **Tool** page, enter the **delete** panel.
2. Switch the selector to **comps**.
3. Click **preview empty**.

The **message** bar displays the message: "3 entities are empty."

These are the **mid1**, **mid2**, and **end** components that no longer have elements in them.

4. Click the entity selector, **comps**, once to see a list of the identified empty components.

A complete list of the model's components appears. The empty components are indicated with an activated check box.

5. Click **return** to return to the **delete** panel.
6. Click **delete entity**.

The **message** bar displays the message, "Deleted 3 comps."

Step 8: Delete all the geometry lines in the model.

You should still be in the **delete** panel.

1. Switch the **entity** selector to **lines**.
2. Select **lines >> all**.
3. Click **delete entity**.

The two lines you created earlier are deleted.

4. Click **return** to return to the main menu.

Step 9: Move the component, **geometry**, to the front in the components list.

1. Access the **reorder** panel in one of the following ways:
 - From the **Collectors** menu, select **Reorder** and choose **Components**.
 - From the **Tool** page, go to the **reorder** panel.
2. Click the **comps** selector to see a list of the model's components.
3. On the right side of the panel, click the switch and select **name(id)**. (Switch from **name** to **name(id)**.)

4. Notice the IDs of the components. The ID for shells is **one**, the ID for rigid is **five**, and the ID for geometry is **six**.
5. Select the component, **geometry**.
6. Click **select** to complete the selection.
7. Activate the option **move to: front**.
8. Click **reorder** to apply the reorder function to the component, **geometry**.
9. The **message** bar displays the message, "*The selected collectors have been moved.*"
10. Click the **comps** selector once to review the reordered list of components.
11. Notice the component, **geometry**, is at the top of the list. However, it still has the same ID, **six**.
12. Click **return** to return to the main menu.

Step 10: Renumber the components to be the same as their position in the list.

1. Access the **renumber** panel in one of the following ways:
 - From the **Collectors** menu, select **Renumber** and choose **Components**.
 - From the **Tool** page, go to the **renumber** panel.
2. Go to the **single** sub-panel.
3. Switch the entity selector to **comps**.
4. Click the **comps** selector to see a list of the model's components.
5. On the panel's right side, select **comps >> all**.
6. Click **select** to complete the selection of components.
7. Verify **start with =** is set to **1**.
8. Verify **increment by =** is set to **1**.
9. Verify **offset =** is set to **0**.
10. Click **renumber** to renumber the components.
11. Click the **comps** selector to review the model's component list.
12. Notice the components are numbered according to their position in the list.
13. Click **return** to return to the main menu.

Having components with IDs that do not reflect their position in the model's list of components will not result in errors. However, having components with IDs that do reflect their position in the model's list of components can be helpful for organizational purposes.

Step 11: Create an assembly containing the components, **shells** and **rigid**.

1. Access the **assemblies** panel in one of the following ways:

- From the **Collectors** menu, select **Create** and choose **Assemblies**.
 - From the **Tool** page, go to the **assemblies** panel.
2. Go to the **create** sub-panel.
 3. For **name =**, enter `elements`.
 4. Leave the **card image =** field blank.
 5. Click the **comps** selector to see a list of the model's components.
 6. Select the components **shells** and **rigid**.
 7. Click **select** to complete the selection of components.
 8. Click **create** to create the assembly.
 9. Click **return** to return to the main menu.

Step 12: Create a load collector named **constraints**.

1. Access the **Load collector** panel in one of the following ways:
 - From the **Collectors** menu, choose **Create** and select **Load Collectors Icon**
 - On the **Collectors** toolbar, click **Load Collectors Panel**
2. Go to the **create** sub-panel.
3. For **name =**, enter `constraints`.
4. Click **color** and choose red.
5. Leave the **card image =** field blank.
6. Click **create** to create the load collector.

The **message** bar displays the message: "*The loadcol was created.*"

7. Left-click anywhere in the HyperMesh window (except on a button) to dismiss the message in the **message** bar.
8. Notice that **loadcol constraints** now appears in the **status** bar. The current load collector is **constraints**. Any loads that are created will be organized into this load collector.
9. Click **return** to return to the main menu.

Step 13: Move the model's one constraint into the load collector, **constraints**.

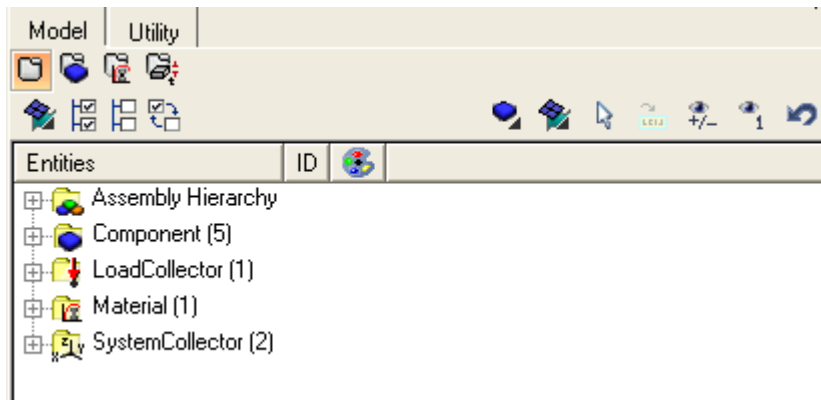
The existing load collector, **loads**, contains several forces and one constraint. The **organize** panel is used to move the one constraint in the load collector, **constraints**.

1. From the **Tool** page, enter the **organize** panel.
2. Go to the **collectors** sub-panel.
3. Switch the **entity** selector to **loads**.
4. Select **loads >> by config**.

5. Click **config =** and select **const**.
6. In the center of the panel, toggle from **displayed** to **all**.
7. Click **select entities**.
8. Verify that **dest =** is set to the load collector, **constraints**.
9. Click **move** to move the **selected (constraints)** into the load collector, **constraints**.

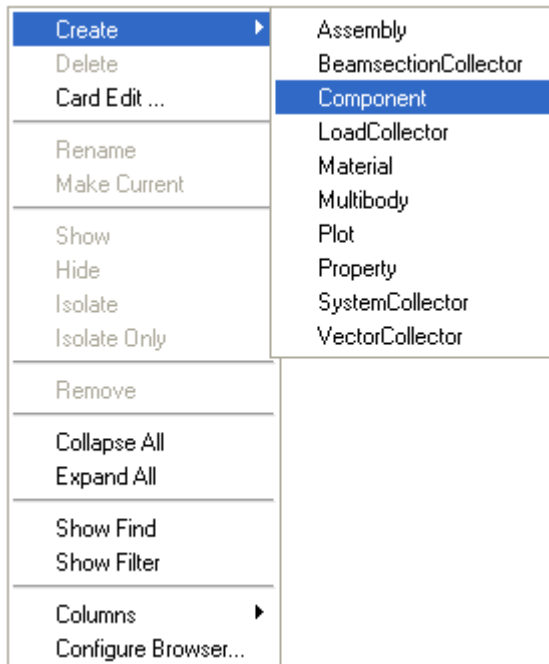
Step 14: Open the *model browser*.

- Click the **Model** tab in the tab area if the tab area is open. Or go to the **View** menu and select **Model Browser**.

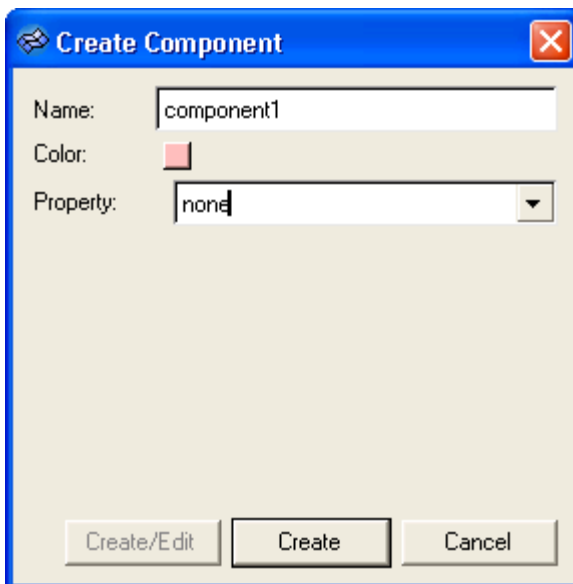


Step 15: Create a component from the *model browser*.

1. Right-click in the white blank area below the list of components, materials, load collectors, and system collectors in the **model browser**.
2. From the pop-up menu, select **Create > Component**.



3. Click the **color** icon and chose the component's color as pink.
4. Click **Create** to create the component.



The component named **component1** is appended to the list.

5. In the **model browser**, click the **+** button beside the **Components** entity to see that **Component1** is bolded in the list to indicate it is the current component.

Step 16: Review the existing assembly elements from the **model browser**.

1. Left-click the **+** button next to **Assembly Hierarchy** then click the **+** button next to **elements** to expand its tree. Notice that it contains two components, **rigid** and **shells**.

Step 17: Add the components, *geometry* and *component1*, to the assembly, *assem_mid*, using the *model browser*.

1. Left-click the component name, ***geometry***, to select it.
2. Press the CTRL key and left-click the component name, ***component1***.
Press the CTRL key and left-click a selected item to deselect it.
3. Left-click any one of the selected components and drag the mouse pointer over the assembly, ***assem_mid***. When ***assem_mid*** is highlighted, release the mouse button.

The selected components are added to the assembly, ***assem_mid***.

Use the SHIFT key and left mouse button to select multiple items in the ***Model Browser*** list at one time. Left-click the first item in the list. Then press the SHIFT key and left-click the last item in the list.

Step 18: Rename the assembly, *assem_mid* to *assem_geom*, from the *model browser*.

1. Right-click ***assem_mid*** and select ***Rename***.
Assem_mid is highlighted and ready for editing.
2. Type `assem_geom` and press ENTER.

Step 19: Delete *component1* from the *model browser*.

1. Right-click ***component1*** and select ***Delete***.
2. Confirm that you wish to delete the component by clicking ***yes*** in the delete confirm dialog.

Component1 is deleted.

Notice that in the list, there is no bolded component name. This indicates there is no current component specified.

Step 20: Set the current component from the *model browser*.

1. Right-click ***shells*** and select ***Make Current***.
The component name is bolded.
2. Notice that the ***header*** bar specifies that that current component is ***shells***.

Section 5: Controlling the Display

When performing finite element modeling and analysis setup, it is important to be able to view the model from different vantage points and control the visibility of entities. You may need to rotate the model to understand the shape, zoom in to view details more closely, or hide specific parts of the model so other parts can be seen. Sometimes a shaded visualization is best, while other times a wireframe visualization is needed to work on details inside the model.

HyperMesh has many functions to help you control the view, visibility, and visualization of entities. This exercise introduces you to these functions.

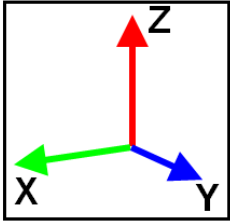
In this section, you will learn how to:

- Control the points of view, mouse, and toolbar.
- Control the visibility of entities using the **display** panel, **mask** panel, and tools on the **Utility** menu.
- Control how entities look by using the toolbar, and the model browser.
- Rename components.
- Identify and delete empty components.
- Delete all the geometry lines.

View control is accomplished through the use of the Toolbar Icons and the mouse.

- **Tool Bar Icons**



Previous View (B)	• Go back to the previous view																															
Fit View (F)	• Fit the model to your screen																															
Plot, Refresh (P)	• Redraw / refresh the graphics area																															
X-Y Plane (Top View)	• X-Y Plane, Drawing View TOP																															
X-Y Plane (Bottom View)	• X-Y Plane, Drawing View BOTTOM																															
X-Z Plane (Left View)	• X-Z Plane, Drawing View LEFT																															
X-Z Plane (Right View)	• X-Z Plane, Drawing View RIGHT																															
Isometric View	• Display model in isometric orientation																															
Reverse Current View	• Reverse model about imaginary vertical axis																															
User Views (V) True View (T)	<div data-bbox="625 1182 1372 1312"> <table border="1"> <tbody> <tr> <td><input checked="" type="checkbox"/> With Mask</td> <td>save1</td> <td></td> <td>restore1</td> <td>left</td> <td>right</td> </tr> <tr> <td></td> <td>save2</td> <td></td> <td>restore2</td> <td>top</td> <td>bottom</td> </tr> <tr> <td></td> <td>save3</td> <td></td> <td>restore3</td> <td>rear</td> <td>front</td> </tr> <tr> <td></td> <td>save4</td> <td></td> <td>restore4</td> <td>iso 1</td> <td>rev</td> </tr> <tr> <td></td> <td>save5</td> <td></td> <td>restore5</td> <td>cw</td> <td>ccw</td> </tr> </tbody> </table> </div>	<input checked="" type="checkbox"/> With Mask	save1		restore1	left	right		save2		restore2	top	bottom		save3		restore3	rear	front		save4		restore4	iso 1	rev		save5		restore5	cw	ccw	
<input checked="" type="checkbox"/> With Mask	save1		restore1	left	right																											
	save2		restore2	top	bottom																											
	save3		restore3	rear	front																											
	save4		restore4	iso 1	rev																											
	save5		restore5	cw	ccw																											
	• User Views: Pre-defined & drawing views • True Views: View based on a vector of angles																															
Circle Zoom (Z)	• Draw a circle (freeform) around the area to be magnified																															
Dynamic Zoom (S)	• Zoom in or out by dragging the mouse vertically																															
Zoom In (+)	• Zoom into the center of the screen																															
Zoom Out (-)	• Zoom out																															

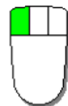


Dynamic Rotate (A) Dynamic Spin (R)	<ul style="list-style-type: none"> •Manually rotate by “grabbing” a point and dragging •Rotate relative to the mouse cursor and screen center
Pan Center (C)	<ul style="list-style-type: none"> •Pan the model •Select and click on the screen to re-center model in graphics window
Rotate Left Rotate Right	} Incremental rotations in respective directions
Rotate Up Rotate Down	
Rotate Clockwise Rotate Counter Clockwise	

- **Mouse Control** – The preferred method for Display Control is the use of the Mouse Buttons.

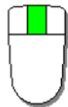
With the CTRL key held on the keyboard the mouse provides total control over rotation, zoom and pan.

- Left mouse button



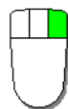
- +CTRL & drag for rotate
- +CTRL & click on entity to change center of rotation
- +CTRL & click in graphics area, off entities to reset center to middle of screen

- Middle mouse button



- +CTRL & drag for zoom
- +CTRL & click for fit

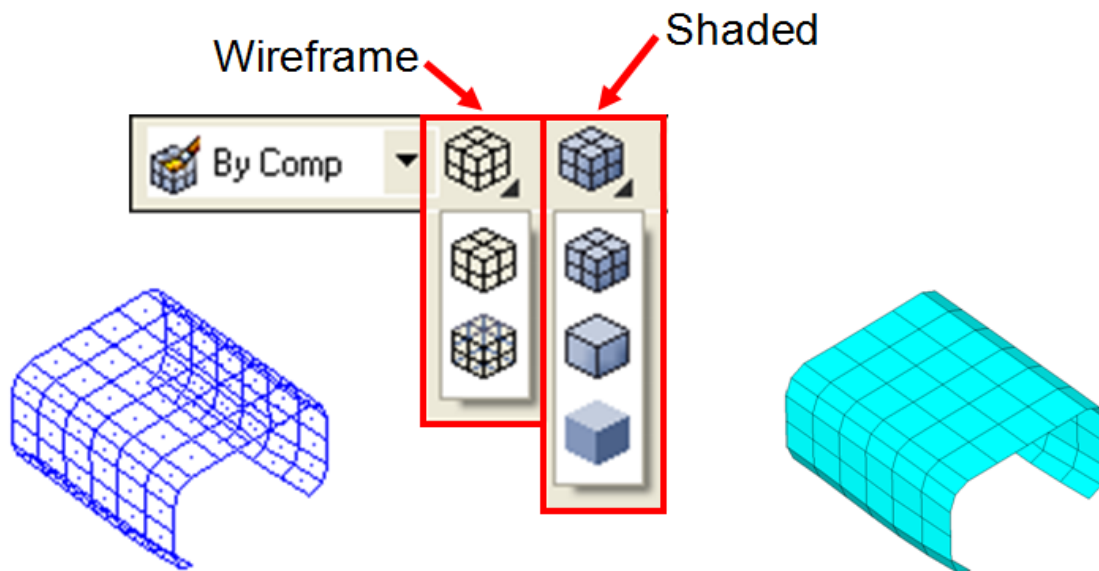
- Right mouse button



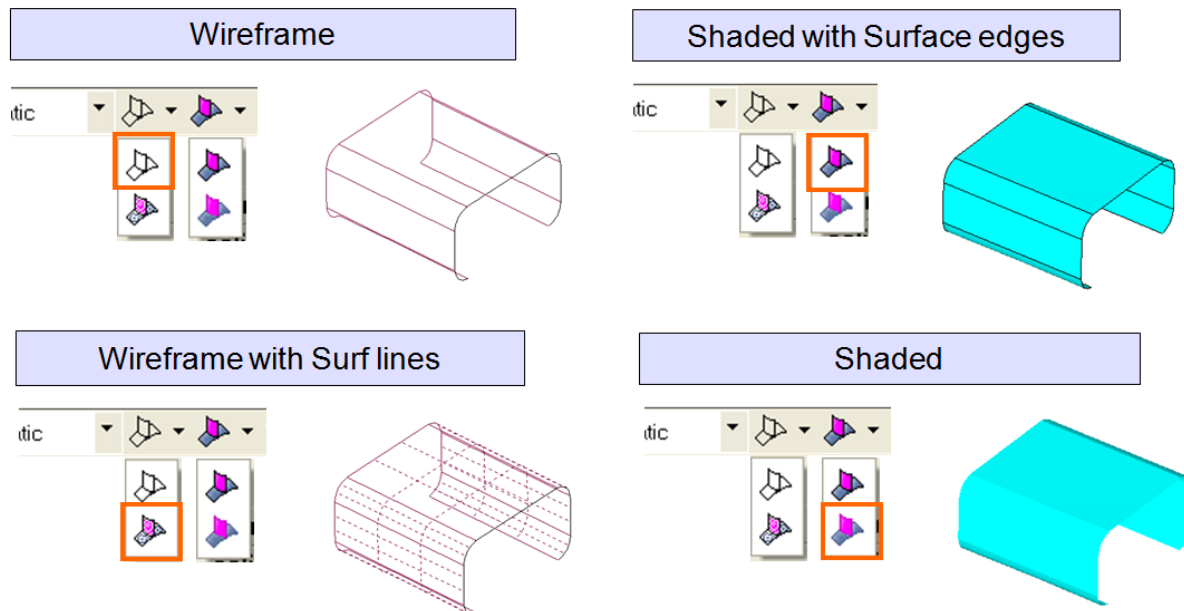
- +CTRL & drag for pan

Model Visualization

- Elements



- Geometry



Mask

The masking tools allow the user to show and hide select entities that might interfere with desired visualization. The icons can be found on the Display toolbar and are used as follows;



MASK – Allows for the selection of entities to hide

REVERSE – Flips the display of masked and unmasked entities.

UNMASK ADJACENT – Unmasks entities immediately adjacent to those shown.

UNMASK ALL – Unmasks all entities.

MASK NOT SHOWN – Masks all the entities that are outside the viewing area.

SPHERICAL CLIPPING – Allows for a sphere of visualization to be defined.

FIND – Allows for elements to be found (typically by ID)

DISPLAY NUMBERS – Utility to turn on and off the display of entity numbers.



DISPLAY ELEMENT HANDLES – Turns on and off element handles.

DISPLAY LOAD HANDLES – Turns on and off load handles.

DISPLAY FIXED POINTS – Turns on and off fixed points.

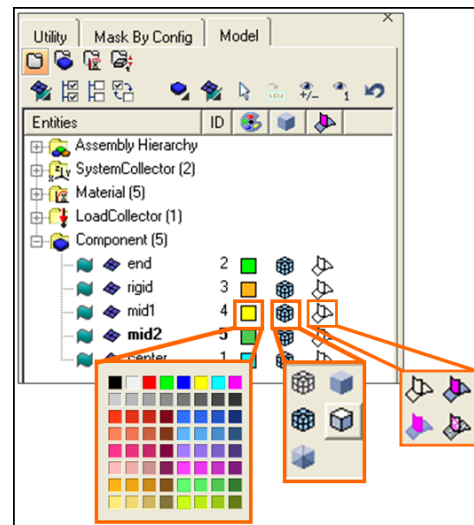
Model Browser

The model browser is a powerful tool for controlling the visualization of the model. In the model browser the user can;

- Turn on and off the display of the geometry  and elements  of collectors



- Control the color of the collector (Right Click)
- Control the visualization of the collector. (Right Click)



Exercise: Viewing a Model

Step 1: Retrieve the HyperMesh model file, `bumper.hm`.

Step 2: Manipulate the model view using the mouse controls.

The CTRL + mouse keys are used to rotate the model, change the center of rotation, zoom, fit, and pan.

1. Move the mouse pointer into the graphics area.
2. Press the CTRL key + left mouse button and move the mouse around.

The model rotates with the movement of the mouse.

A small white square appears in the middle of the graphics area, indicating the center of the rotation.

Release the left mouse button and press it again to rotate the model in a different direction.

3. Press the CTRL key and quick-click the left mouse button anywhere on the model.

The center of rotation square appears near where you clicked.

HyperMesh searches for one of the following conditions in the listed order and relocates the center of rotation at or near the first condition identified (if none of the conditions are met, the center of rotation is relocated to the center of the screen):

- A nearby node or surface vertex
- A nearby surface edge to project onto
- A nearby geometry surface or shaded element

4. Press the CTRL key + left mouse button to rotate the model and view the change in rotation behavior.
5. Press the CTRL key and quick-click the left mouse button anywhere in the graphics area, except for on the model.

The center of rotation square is relocated to the center of the screen.

6. Press the CTRL key + left mouse button to rotate the model and observe the change in rotation behavior.
7. Press the CTRL key + middle mouse button, move the mouse around, and then release the mouse button.

A white line is drawn along the path of the mouse movement. When the mouse button is released, HyperMesh zooms in on the portion of the model where the line was drawn.

You can also simply draw a line to zoom in on a portion of the model.

8. Press the CTRL key + quick-click the middle mouse button.

The model is fitted to the graphics area.

9. Press the CTRL key and spin the mouse wheel.

The model zooms in or out depending on which direction you spin the mouse wheel.


10. Move the mouse pointer to a different location in the graphics area and repeat #9.

Notice the model zooms in or out from where the mouse handle is located.

11. Press the CTRL key + quick-click the middle mouse button to fit the model to the graphics area.
12. Press the CTRL key + right mouse button and move the mouse around.

The model is panned (translated) according to the mouse movement.

Step 3: Manipulate the view of the model using the rotate functions on the toolbar.

1. On the View Controls toolbar, left-click **dynamic rotate**, .

The **status** bar displays the message, "Move the mouse into the graphics region."

2. Move the mouse pointer into the graphics area.

The center of rotation square appears.


3. Press and hold the left mouse button, and then move the mouse around.

The model rotates with the movement of the mouse, similar to the way the model rotates when you press the CTRL key + left mouse button and move the mouse.

4. Click the middle mouse button on the model.

The center of rotation square appears near where you clicked.

5. Move the mouse pointer out of the graphics area or right-click to exit the rotation mode.

6. On the View Controls toolbar, right-click **dynamic rotate**, , and move the mouse pointer into the graphics area.

Again, the center of rotation square appears. You can click the middle mouse button on the model to change the center of rotation.

7. Press the right mouse button near the center of rotation square.

The model rotates continuously in the direction of your mouse pointer, relative to the center of rotation.

8. With the right mouse button still pressed, move the mouse around slowly.

The direction and speed of the rotating model changes. The farther the mouse pointer is from the center of rotation, the quicker the model rotates.

You can release the right mouse button, and then press it again to rotate the model in a different direction.

9. Middle mouse click anywhere in the graphics area, except on the model.

The center of rotation square is relocated to the screen's center.

10. Move the mouse pointer out of the graphics area or left-click to exit the rotation mode.

Step 4: Manipulate the view of the model by using the zoom in and out functions on the *toolbar*.

1. On the View Controls toolbar, left-click **circle / dynamic zoom**, .

The **status** bar displays the message, "Circle the data to be zoomed in on."

2. Move the mouse pointer into the graphics area.
3. Press the left mouse button, move the mouse around, and then release the left mouse button.

A white line is drawn along the path of the mouse movement. When the mouse button is released, HyperMesh zooms in on the portion of the model where the line was drawn.

You can also simply draw a linear line to zoom in on a portion of the model.

This function is similar to pressing the CTRL key + middle mouse button to zoom into a portion of the model.

4. On the **Standard Views** toolbar, click **fit**, .

The model is fitted to the graphics area.

5. On the **View Controls** toolbar, left-click **zoom in / out**, .

The model is zoomed in by the factor specified in the options panel.

6. On the **View Controls** toolbar, right-click **zoom in / out**, .

The model is zoomed out by the same factor.

7. From the **Preferences** menu select **Meshing Options** or **Geometry Options**.

8. Go to the **geometry** or **mesh** sub-panel.

9. For **zoom factor** =, specify **4**.

10. Click **return** to return to the main menu.

11. On the **View Controls** toolbar, left-click **zoom in / out**, .

The model is zoomed in by the larger, specified factor.

12. On the **View Controls** toolbar, right-click **circle / dynamic zoom**, .









The **status** bar displays the message "Move the mouse into the graphics region."

13. Move the mouse pointer into the graphics area, press the right mouse button, and then move the mouse pointer up and down.

The model is zoomed in and out according to how far you move the mouse up or down.






14. Move the mouse pointer out of the graphics area or left-click to exit the dynamic zoom mode.

Step 5: Manipulate the model view using the arrows and *view* panel on the toolbar.



- On the **View Controls** toolbar, right-click or left-click any of the **rotate** icons (, , ).
The model rotates in the direction of the arrow by the rotation angle specified in the options panel.
- On the **Standard Views** toolbar, left-click **view**, .
The **view** panel pops-up.
- Click **top** to view the model along the Z-axis.
- Exit the **view** panel by moving the mouse pointer out of the pop-up window.
- From the Preferences menu select Meshing Options or Geometry Options.
- For **rotate angle** =, specify a value of **90**.
- Click **return** to return to the main menu.
- Click either **rotate** icons (, , ).
Notice the model rotates by the new specified rotation angle, **90**.
- Change the view of the model to any view.
- Use CTRL + left mouse button, or the **rotate** icons on the **toolbar** to rotate the model.
- Use CTRL + middle mouse button, or the **zoom** icons on the toolbar to zoom into or out on the model.
- On the **Standard Views** toolbar, left-click **view**, .
- For **save1** = enter my view.
- Click **save1** = to save the view.
- Click **top** to display a different view of the model.
- Click **restore1** to retrieve the view stored for **save1** =.
Both the view angle of the model and level of zoom are saved.
- Move the cursor away from the **view** panel to exit.

Step 6: Control the display of components using the *toolbar*.

- On the **Visualization** toolbar, click **Shaded Elements & Mesh Lines**, .
- Notice the shell elements now have been shaded.

3. Right-click **Shaded Elements & Mesh Lines**, , and switch to **Shaded Elements & Feature Lines**, .
4. Notice now the elements shading does not show any mesh lines. Only feature lines are displayed.
5. Right-click **Shaded Elements & Feature Lines**, , to access **Shaded Elements**, .
6. Notice now the feature lines are also removed from the display.
7. Click **Wireframe Elements (Skin Only)**, , to return to the wireframe shading mode.

Step 7: Control the display of components using the *Visual Attributes* panel.

1. On the **Visualization** toolbar, click **Geom/Mesh Styles**, .
2. Click **hidden line**, .
3. Select a component from the list of components.
The component's elements are displayed in hidden line (shaded) mode.
4. Click **all** to select all components and set their display mode to hidden line (shaded) mode.
5. Experiment with the other display modes.

Icon Display Mode



Wireframe – Element edges are displayed with lines.



Hidden Line – The element is displayed as a filled polygon.



Hidden Line with Mesh Lines – The element is displayed as a filled polygon with the feature edges drawn in mesh line color.



Hidden Line with Feature Lines – The element as a filled polygon with the feature edges in mesh line color.



Transparent – The element is displayed as a filled transparent polygon.

6. Click **return** to return to the main menu.

Step 8: Control the visibility of various entity categories using the *model browser*.

1. In the **tab** area, activate the **Model** tab.
2. Right-click anywhere in the white space of the browser list and pick **Expand All** from the context menu.

This will open all of the branches of the **model browser**.

3. At the top of the **model browser**, click **Display none**, .

The display of all entities in the model is turned off.

4. Click **Display all**, .

All entities in the model are now displayed.

5. In the browser list, left-click **Component (5)** to select it.

6. Click **Display none**, .

The display of component collectors is turned off, but all other entities remain displayed.

Display all, **Display none**, and **Display reverse** act globally (on all entities) if nothing in the browser list is selected. If a folder is selected (highlighted), the action will be performed only on the entities within that folder. If an individual entity is selected, the action will apply only to that entity.

7. Left-click in the white space of the browser list.

This deselects all entities in the browser list.

8. Click **Display reverse**, .

The display is reversed. Only the components are now shown instead of everything but components.

9. Click **Component view**, .

Only the component collectors are shown in the browser list.

10. Right-click the **elements and geometry filter** to change it from **Elements + Geometry**, , to **Elements**, .

Display all, **Display none**, and **Display reverse** will no longer affect the display of geometry in the components.

11. Click **Display none**, .



Only geometry is now left in the display.

12. Switch the **elements and geometry filter** back to **Elements + Geometry**, .

13. Click **Display reverse**, .

Only elements are now displayed.


Step 9: Control the visibility of individual components using the *model browser*.

1. Click **elements**, , next to the **mid2**, **end**, and **rigid** components.
2. Only the components **center** and **mid1** have their elements displayed now.
3. Press F on the keyboard.
4. The displayed components are fitted to the graphics area.
5. Click **geometry**, , next to the components **mid2** and **end**.
6. Fit the displayed components to the graphics area (F).


The geometry in the components, mid2 and end, and the elements in the components, center and mid1, are displayed.


A component collector has two 'compartments': one for elements and the other for geometry. The display for elements and geometry in a component can be controlled separately.

Step 10: Control the display of entities using the *mask* panel.

1. Access the **mask** panel in one of the following ways:
 - On the **display** toolbar, click **Mask**, 
 - From the main menu, click the **Tool** page, then select the **mask** panel
2. Go to the **mask** sub-panel.
3. With the **elems** selector active, select **elems >> by collector**.
4. Select the component, **mid1**.
5. Click **select** to complete the selection of components.
6. From the graphics area, manually select a few elements in the **center** (blue) component.
7. Click **mask** to mask the elements.

The elements in the mid1 component and the elements you selected from the graphics area are no longer displayed.

8. In the model browser, notice that the **elements** () for the components center and mid1 are still displayed. Their display icons indicate that they are activated even though some or all of the elements in these components are masked (hidden).

9. In the **mask** panel, click **unmask all**, or on the **display** toolbar click **unmask all**, .


All the elements in the components, **center** and **mid1**, are visible again.

Notice the elements in the other components are not displayed. This is because these components are not active in the Display panel.

10. Click **return** to exit to the main menu.

Step 11: Control the display of entities using the *find* panel.

1. Access the find panel in one of the following ways:

- On the **display** toolbar, click **find**, .
- From the main menu, click the **Tool** page, then select the **find** panel.


2. Go to the **find entities** sub-panel.

3. Select **elems >> by collector**, and select the component, **end**, from the components list.

4. Click **select** to complete the selection of components.

5. Click **find** to find the elements.

The elements in the component, **end**, are displayed.


5. In the model browser, notice that the elements for the component, end, are now shown as **active** (). This is because the collector containing the entities that are to be displayed (found) must be active.

6. Go to the **find attached** sub-panel.


7. For **attached to:**, select **elems >> displayed**.

8. Click **find** to find the elements.

Some of the elements in the components, mid2 and rigid, are now displayed. These elements are immediately adjacent and connected to the selected elements.

Again, notice that the elements for these components are now shown as **active** (). The components were made active so that the elements could be displayed.

9. Click **return** to return to the main menu.

10. On the **display** toolbar, click **Unmask All** (.

All of the model's elements are now displayed. This is because the find panel finds the entities it is supposed to find, activates (displays) the corresponding collectors, and masks the other entities in the collectors it activated. In this case, the last find command displayed on the components, mid2 and rigid, in the model browser.

Step 12: Change the display of entities using the *Mask By Config* tab.

1. In the **model browser**, click in the white space of the browser list to make sure nothing is selected.

2. Click **Display none** (.

3. Click **Display all** (.

Performing these two steps makes sure that everything is displayed in the model.

4. Click to the **Mask By Config** tab.

5. Click the **1** in the **Isolate** column for **Components**.

Only the components in the model (elements, geometry, and connectors) are displayed. Everything else is now masked.

6. Expand the **Components** branch to expose connectors, elements, and geometry.

7. Click the – in the **Hide** column for **Elements**.

The elements in the model are now masked, leaving only surfaces shown.

8. Expand the **LoadCollectors** branch to expose loads and equations.

9. Expand the **Loads** branch to expose constraints, forces, moments, etc.

10. Click the + in the **Show** column for **Constraints**.

Constraints are added to the display of the surfaces.

11. Expand the **Elements** branch to expose **0D/rigids**, **springs/gaps**, **1D**, **2D**, and **3D**.


12. Click the **1** in the **Isolate** column for **0D/Rigids**.

Rigid elements are added to the display, while the surfaces are now masked. The constraints remain displayed. This is because using isolate below the top level of the list will not mask anything outside of the top level that entity belongs to. Rigids belong to components at the top level, so no entities in the other top levels are masked.

13. Click the **1** in the **Isolate** column for **Components**.

All of the entities in the components are shown again, while the constraints are masked. This is because using isolate at the top level of the list (components, groups, loadcollectors, morphing, multibodies, and systemcollectors) will mask everything outside of the entity type being isolated.

Step 13: Change the color of components using the *model browser*.

1. Click the **Model** tab to go to the *model browser*.
2. Right-click **color** () next to **mid2**.
3. From the color pop-up, select a different color.



4. Observe the change in color of the elements in **mid2**.

Step 14: (Optional) Save your work.

Chapter 2

Geometry Clean-up

Section 1: Importing and Repairing CAD

HyperMesh is capable of importing geometry from many CAD sources. Most of the popular CAD packages are read directly and for those that are not HyperMesh supports the popular intermediate languages as well. While the importation of data generally occurs with little error, there are issues that can occur, and as such HyperMesh, offers a wide variety of tools to remedy these geometric issues.


The benefits of importing and repairing CAD are:

- Correct any errors in the geometry from import
- Create the simplified part needed for the analysis
- Mesh a part all at once
- Ensure proper connectivity of mesh
- Obtain a desirable mesh pattern & quality

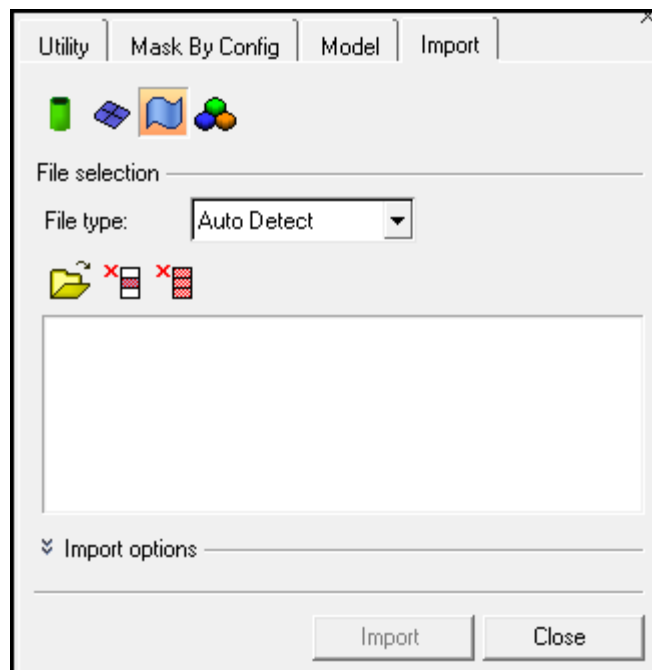
In this section, you will:

- Delete untrimmed surfaces
- Close missing surfaces
- Set the cleanup tolerance
- Equivalence free edges
- Delete duplicate surfaces

Geometry Import

Importing geometry occurs in the import tab, accessible through the Import Icon . Using this tab the user can import data from popular CAD packages such as

- **Unigraphics (NX2,NX3,NX4,NX5)**
 - Supports import of .prt files
 - Provides a UG part browser
 - Requires an installation of UG to be accessible, either locally or on a network
- **CATIA (V4,V5)**
 - Supports import of .model (V4) files
 - Additional license from Altair is required of .catpart (V5) file import.
- **Pro/Engineer (Wildfire 2.0 & 3.0)**
 - Supports import of .prt and .asm files.

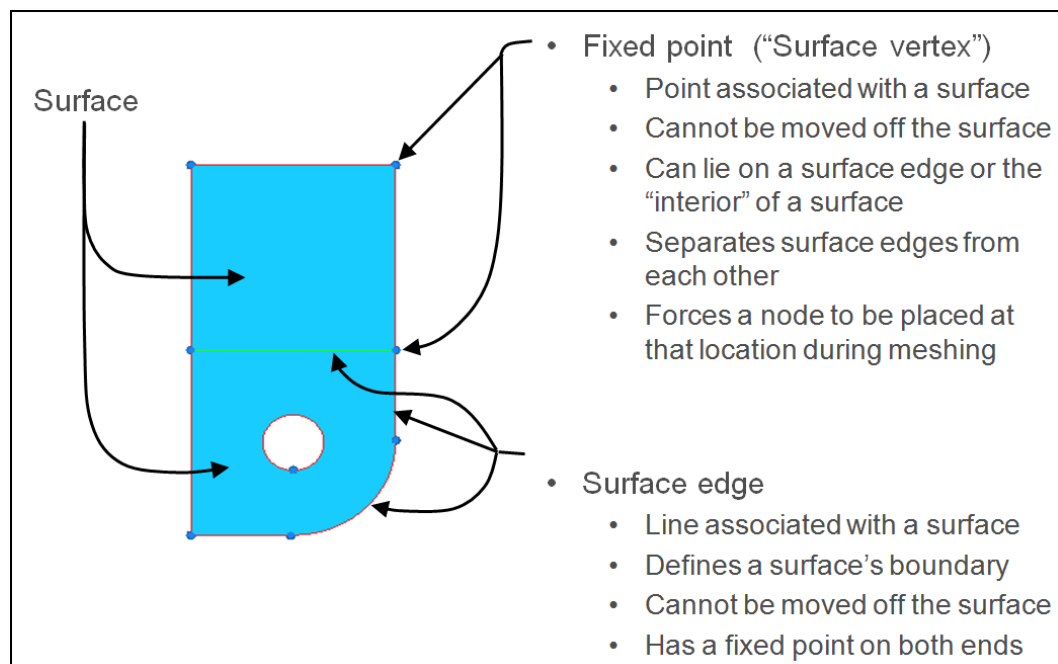


Additionally HyperMesh supports the import of the following intermediate translational languages:

- **IGES (.igs & .iges)**
- **STEP (.stp)**

Topology Repair

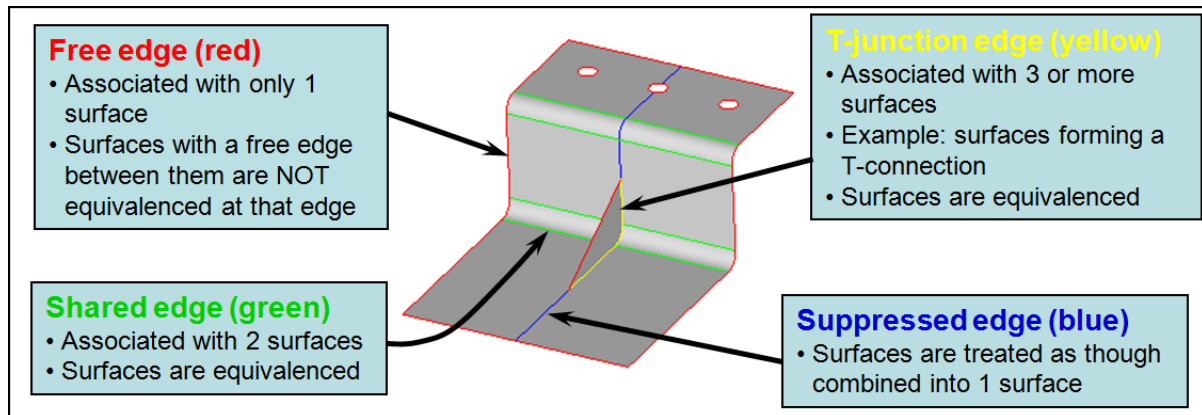
Surface Definition



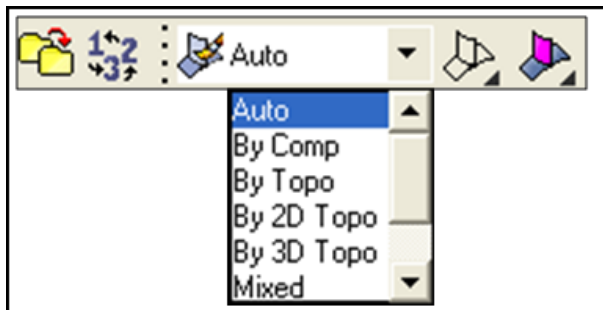
Topology

Topology deals with the connection of adjacent surfaces. Connectivity is critical as HyperMesh will create a contiguous mesh over connected faces thus guaranteeing stresses, strains and deformations will propagate over the part in a realistic manner. When importing native CAD files, HyperMesh will read surface connectivity directly from the file. When using intermediate languages (IGES or STEP) HyperMesh will use a tolerance calculation to determine when two edges should be connected. While this will work very well, there can be issues with connectivity that will need to be fixed before meshing.

HyperMesh allows for easy visualization of surface connectivity through the use of an edge color scheme shown below;




Topology Visualization

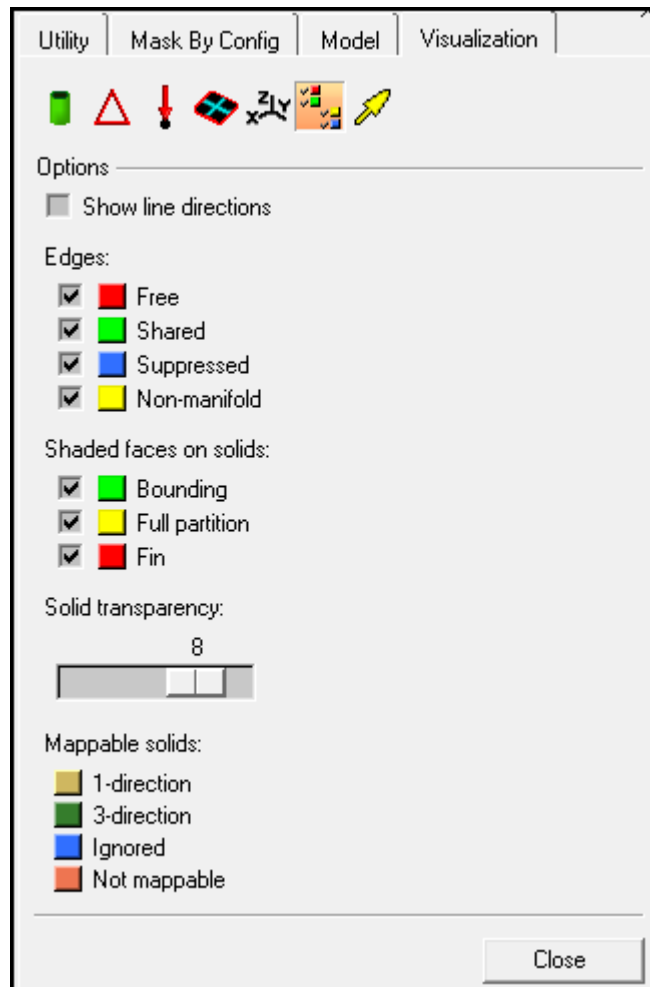


Display of topology can be controlled with this pulldown found on the icon tool bar.

- **Auto:** Surfaces and Edges will be displayed in the component color unless the user is in a topological editing function where then surfaces will be colored grey and edges will be in their topological color.
- **By Comp:** Surfaces and Edges will always be displayed in component colors.
- **By Topo:** Surfaces will always be colored grey and edges will always be in their topological color.
- **By 2D Topo:** Only 2D topology will be displayed in topological color.
- **By 3D Topo:** Only 3D topology will be displayed in topological color.
- **Mixed:** Similar to By Topo but while the edges will be in topological colors, the surfaces will be in their component color and not grey.

Topology Visualization cont







On the toolbar the Topology Options Icon  will open the visualization tab



This tab will allow the user to display or hide 2D and 3D topology based on its type.

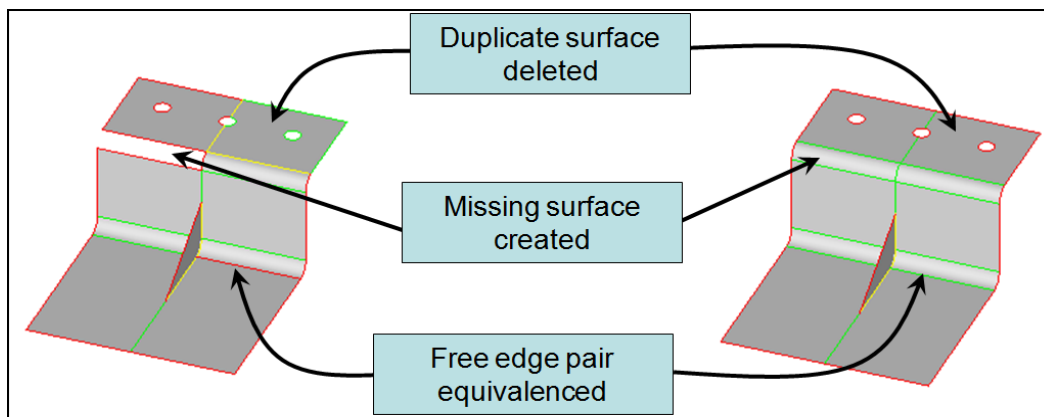
This tab also allows the user to control the transparency and change the shading colors of mappable solid regions.

Other functionality in this tab allows visualization control over the following:

- Connector 
- Constraints 
- Loads 
- Morphing 
- Systems 
- Vectors 

Topology Repair

While HyperMesh will in most cases create proper and connected geometry accurately representing the initial CAD geometry, there can be issues with the import. Common issues are:

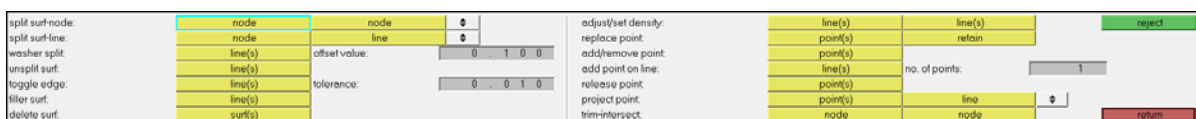


Topology Repair: Tools

HyperMesh has a supply of tools to repair issues in the geometry.

The ultimate goal of Topology Repair is to assure that the geometry that is used to create the mesh is as close as possible to the design intent of the original CAD data. To this end, these are the tools that HyperMesh provides;

- **Edge Edit Panel**
 - **Equivalence (multiple edges at a time)**
 - Search surfaces for pairs of free edges and combine into shared edges
 - **Toggle (1 edge / edge pair at a time)**
 - Select an edge; equivalences with other free edges found within a user specified tolerance
 - **Replace (1 edge pair at a time)**
 - Select 2 edges to equivalence together
 - Control which edge to retain and which to move
- **Point Edit Panel**
 - **Replace** (1 edge at a time)
 - **Release** – Combine pairs of free edges with gaps between them into shared edges
- **Defeature Panel**
 - **Duplicates** – Identify and delete duplicate surfaces within a user specified tolerance
- **Surfaces Panel**
 - **Spline / filler** – Select lines / surface edges to recreate any missing surfaces
- **Quick Edit Panel**



The Quick Edit Panel is a “tool box” of utilities for geometry repair. Many of the tools can be found in other panels and their functionality is exactly the same. The Quick Edit panel simply provides a single location for many of the most often used tools. These **include**:

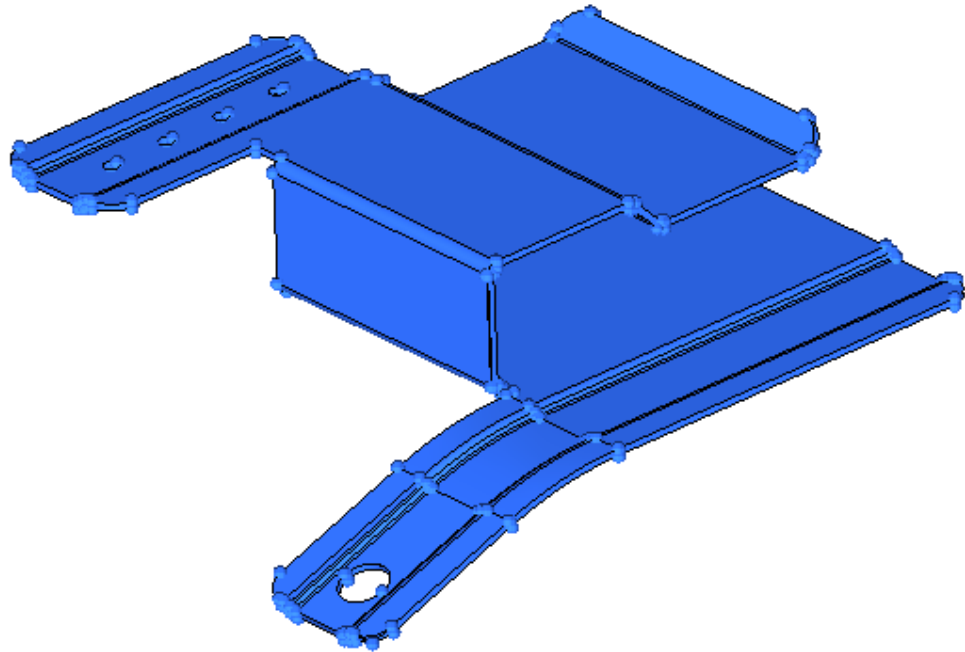
- **Unsplit** – Removes / deletes an edge created by splitting a surface in HyperMesh.
- **Toggle** – Same as edge edit panel; change edge type within tolerance.
- **Filler surf** – Select a line on a free surface edge to recreate any missing surfaces.
- **Delete surf** – Same as delete panel (surfaces only).
- **Replace point** – Same as point edit panel; move/retain point.
- **Release point** – Same as point edit panel; must be associated with line.

Topology Repair: Strategy

- ✓ Understand the size and scale of the model
 - With models that represent everything from full size ships to microscopic electronic parts all residing in a graphics area on a computer monitor, it is often difficult to understand the overall scope of the model. It is critical to get an idea of the overall size of the model and determine a global element size that will be applied to the eventual mesh.
- ✓ Set a cleanup tolerance based upon the previously determined global element size.
 - With the element size established, a cleanup tolerance can now be set. The cleanup tolerance specifies the largest gap size to be closed by the topology functions. This value should never exceed 15-20% of the global element size. Values beyond this limit can introduce distortion into the mesh.
- ✓ Use topology display tools to determine what needs to be fixed.
- ✓ Find duplicate surfaces and delete them.
- ✓ Use **equivalence** to combine as many free edge pairs as possible.
 - Visually verify no surfaces were collapsed with this function.
- ✓ Use **toggle** to combine and remaining edges.
 - Use replace if more control is needed.
- ✓ Use **filler surface** to fill in any missing surfaces.

Exercise: Importing and Repairing CAD Geometry Data

This exercise uses the model file, `clip_repair.hm`.



Step 1: Open and view the model file, `clip_repair.hm`.

Step 2: View the model in topology display toolbar and shaded mode to evaluate its integrity.


1. Observe where the model has incorrect connectivity and missing or duplicate surfaces.
2. Go to the **autocleanup** panel.

Note that the surface edges are now colored according to their topology status. This

occurs because **Geometry Color** is set to **auto** ().

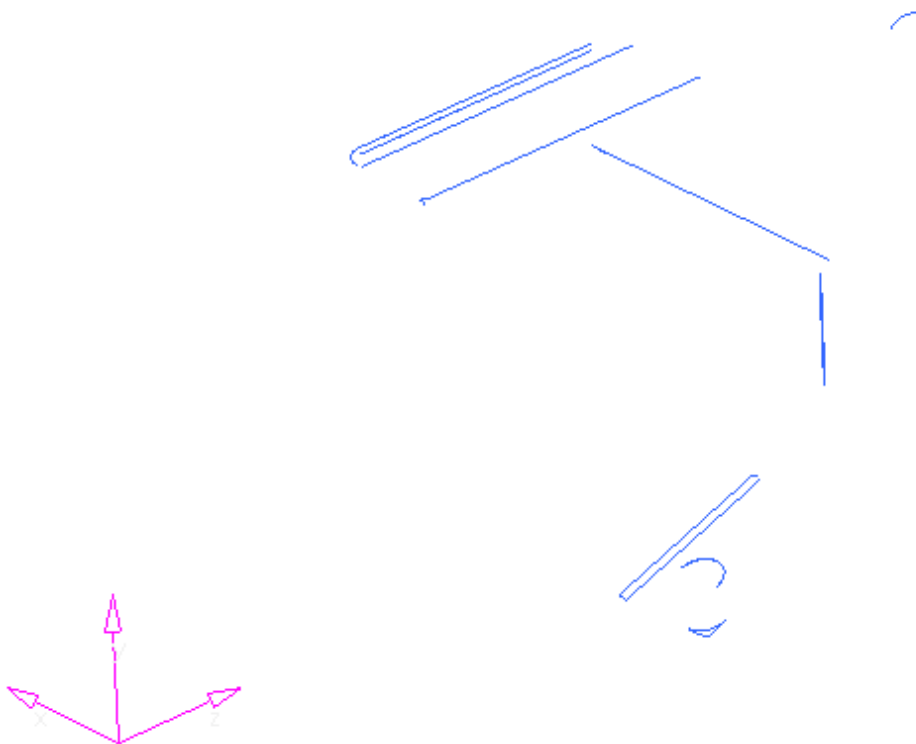
3. Click **Wireframe Geometry** () and **Shaded Geometry & Surface Edges** () to explore the different display modes.

The toolbar contains icons that control the display of the surfaces and surface edges. Surfaces can be shaded (with or without edges) or displayed in wireframe. Right-click the icons to access the drop-down menu for additional options. Place your mouse over the cursor to view a description of the button's functionality.



4. Click **Visualization** () and navigate to the **Topology** tab.

Visualization controls the display of the surfaces and surface edges. Surfaces can be shaded or wireframe. The check boxes within this menu turn the display of the different edge types and fixed points (surface vertices) on or off.

5. Clear all the check boxes except the **Free** check box.
6. Move the mouse cursor off the pop-up menu to close it.
Only the free edges should be displayed at this point.
7. Observe the free edges and make note of where they are.
The free (red) edges show where there is incorrect connectivity or gaps.
8. Note the locations where there are closed loops of free edges. These are locations that probably have missing surfaces.



Free edges indicating surface discontinuities of the clip geometry

9. Click the **Visualization** icon, , and select only the **T-junctions edges** check box.
10. Observe the t-junction edges and make note of where they are.
The t-junction edges show where there are more than two surfaces sharing an edge, which might be incorrect connectivity. For this part, there are yellow edges completely surrounding two areas. This indicates that there are probably duplicate surfaces in these locations.
11. Click the **Visualization** icon, , and select all the check boxes.

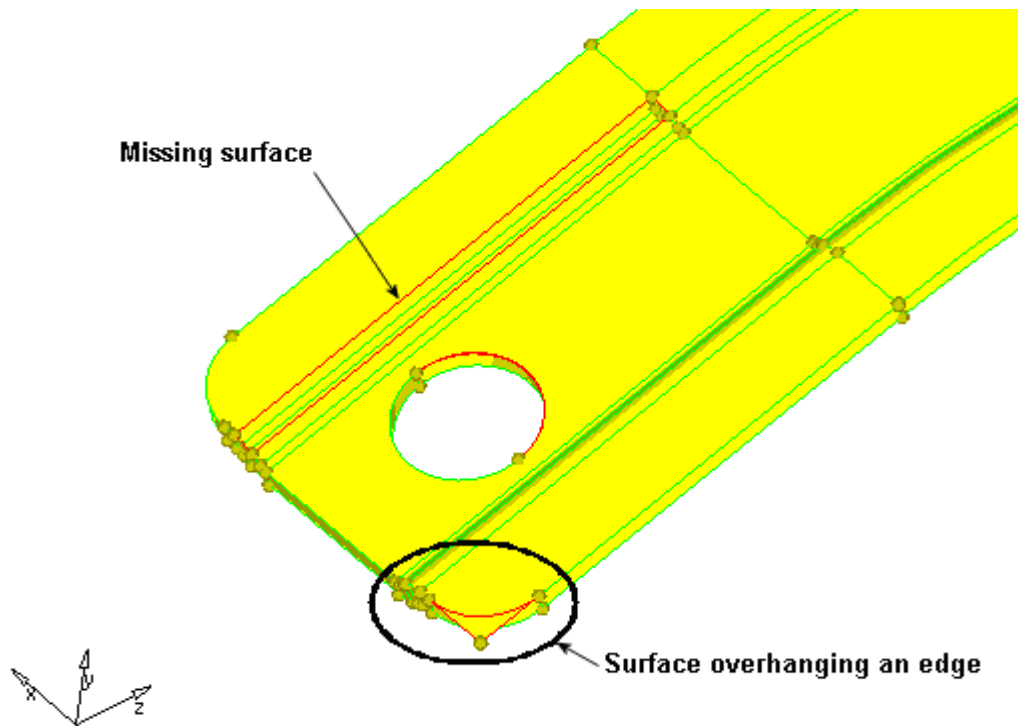
12. Click the **Shaded Geometry & Surface Edges icon**, .

The surfaces should now appear solid rather than having only their edges displayed.

13. Rotate, zoom, and pan to locate any errors in the geometry.

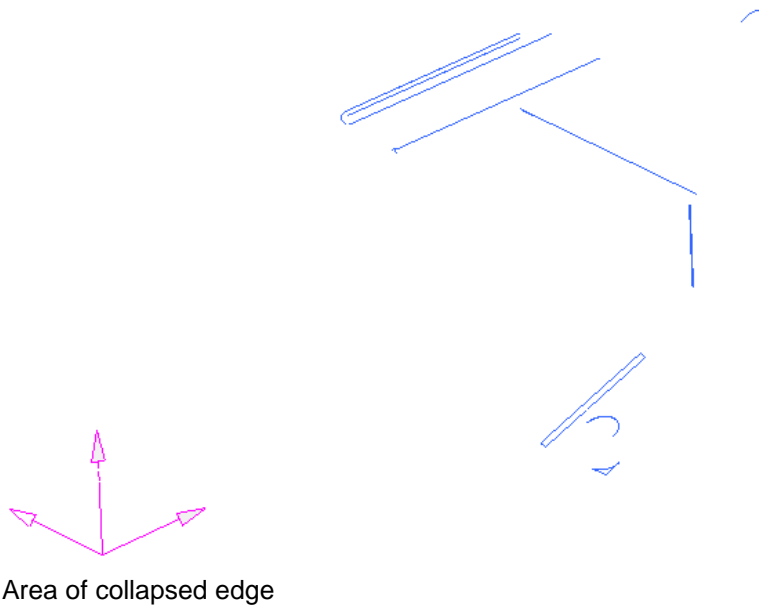
14. Make note of the areas to be worked on. There is:


- A surface that overhangs a round corner
- A missing surface



Surface overhanging an edge and a missing surface

An edge that has apparently been collapsed



15. Click **Wireframe Geometry** () to change back to wireframe.

Step 3: Delete the surface that overhangs the round corner.

1. Enter the **Delete** panel in one of the following ways:
 - From the **Geometry** menu, point to **Delete** and click **Surfaces**
 - Press F2
2. In the **Graphics** area, select the overhanging surface shown in the previous figure.
3. Click **delete entity** and click **return**.
4. **Return** to the main menu.

Step 4: Create surfaces to fill large gaps in the model.

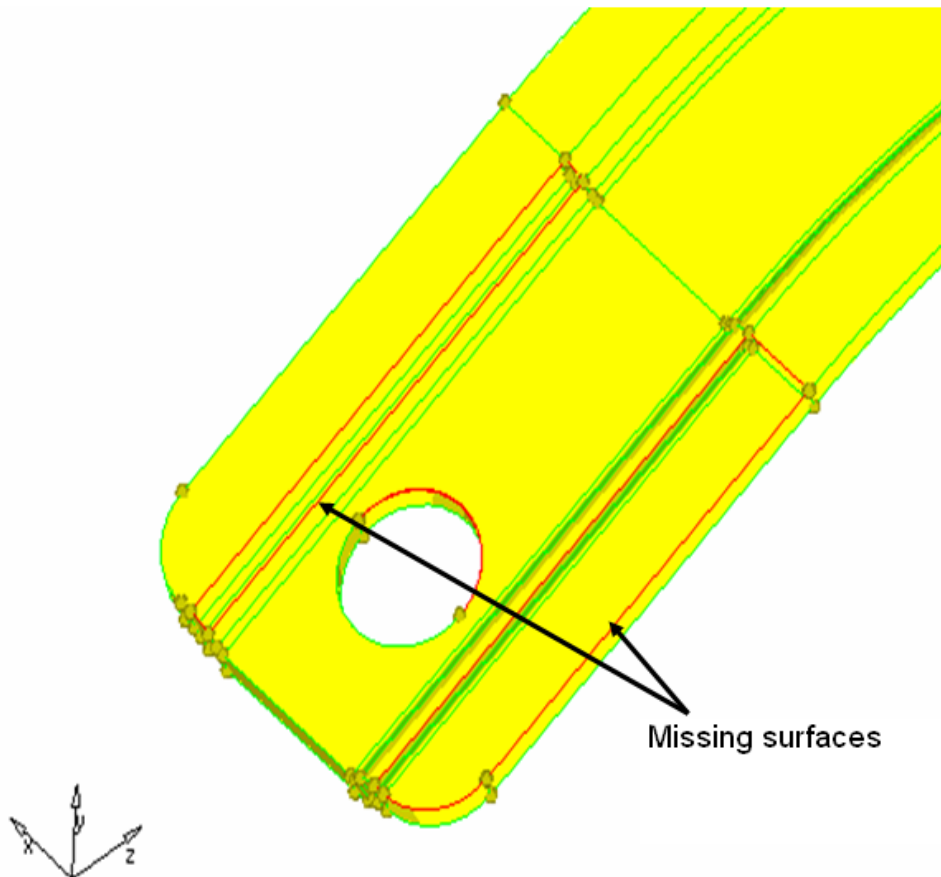
1. Access the **surfaces** panel.
2. Access the **spline/filler** sub-panel.
3. Clear the **keep tangency** check box.

The keep tangency option looks at surfaces attached to the selected edges and tries to create a surface tangent to them. This helps to form a smooth transition to the surrounding surfaces.

4. Verify the entity type is set to **lines**.
5. Verify the **auto create (free edges)** check box is selected.

The auto create option simplifies the selection of the lines bounding the missing surface. Once a line is selected, HyperMesh automatically selects the remaining free edges that form a closed loop, and then create the filler surface.

6. Zoom into the area indicated in the following image.



Area of missing surfaces

7. Pick one of the red lines bounding one of the gaps (missing surfaces).
HyperMesh automatically creates a filler surface to close the hole.
8. Repeat #7 to create a filler surface in the other gap.
9. Click **return**.

Step 5: Set the global geometry cleanup tolerance to .01.

1. Press O to go to the **options** panel.
2. Go to the **geometry** sub-panel.
3. In the **cleanup tol =** field, type 0 . 01 to stitch the surfaces with a gap less than 0.01.
4. Click **return** to go to the main menu.

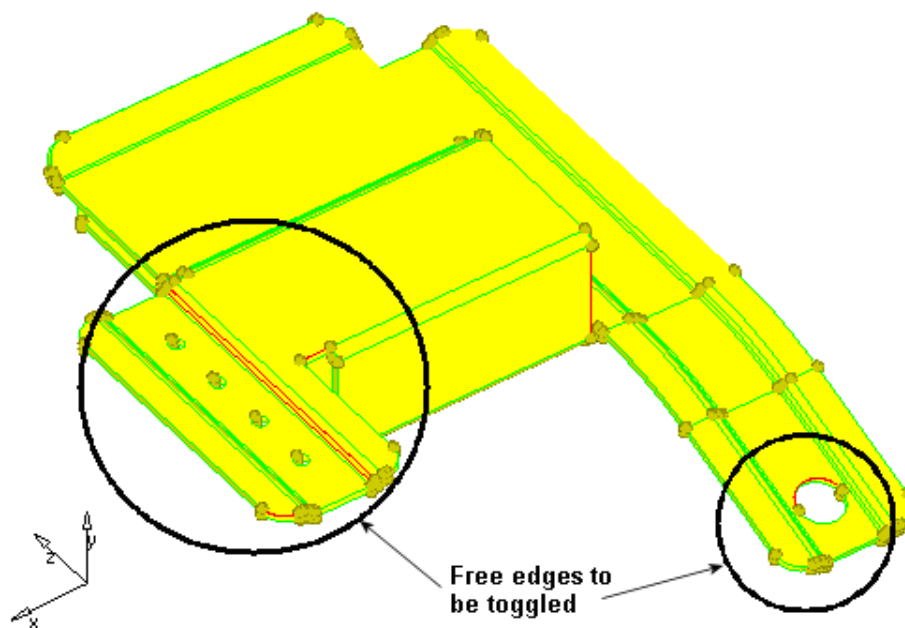
Step 6: Combine multiple free edge pairs at one time with the equivalence tool.

1. Enter the **Edge Edit** panel in one of the following ways:
 - From the **Pull-down** menu, select **Geometry**, then **Edit**, then **edges**
 - From the main menu, select **Geom Page**, then choose **Edge Edit**
2. Go to the **equivalence** sub-panel.
3. Activate the **equiv free edges only** check box.
4. Select **surfs >> all**.
5. Verify that the cleanup **tol=** is set to 0.01, which is the global cleanup tolerance specified in the options panel.
6. Click the green equivalence button to combine any free edge pairs within the specified cleanup tolerance.

Most of the red free edges are combined into green shared edges. The few remaining are caused by gaps larger than the cleanup tolerance.

Step 7: Combine free edge pairs, one pair at a time, using the toggle.


1. Go to the **toggle** sub-panel.
2. In the **cleanup tol =** field, type 0.1.
3. In the **Graphics** area, click one of the free edges shown in the following image.
4. Rotate and zoom into the area if needed. When the edge is selected, it will change from red to green, indicating that the free edge pair has been equivalenced.



Area where free edges need to be toggled

5. Use toggle to equivalence the other edges shown in the image.

Step 8: Combine the remaining free edge pair using *replace*.

1. Go to the **replace** sub-panel.
2. Click **user views** ()
3. Click **restore1** to bring back the saved view.
4. With the selector under **moved edge:** active, click the leftmost free edge in the **Graphics** area.

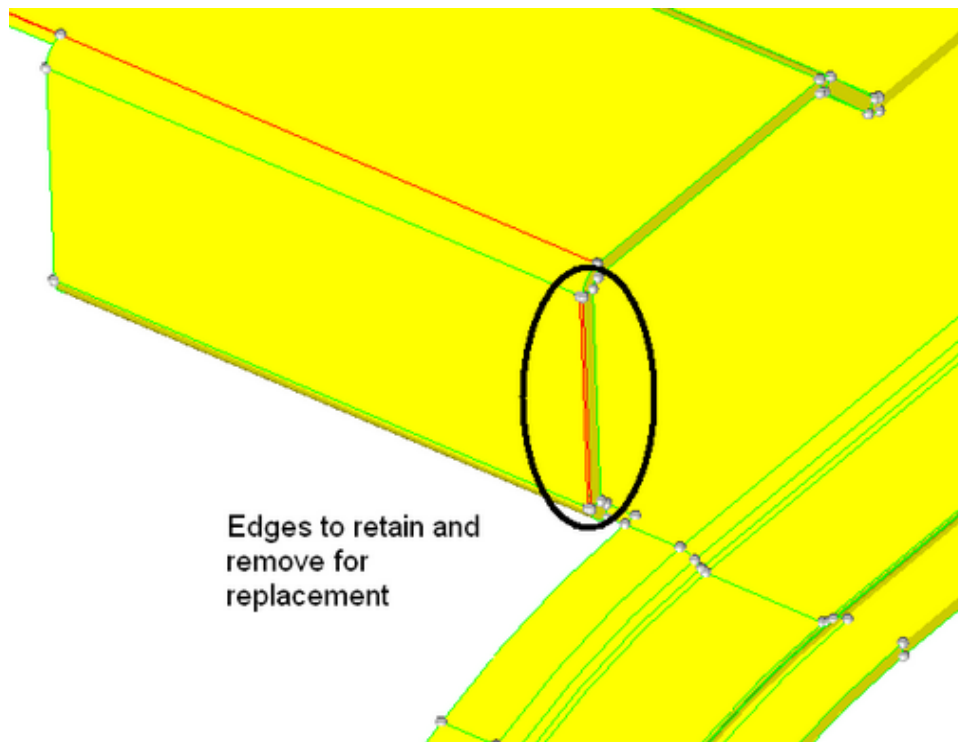
Verify that the selector under **retained edge:** is now active.

5. Select the rightmost red edge.
6. In the **cleanup tol =** field, enter 0.1.
7. Click **replace**.

Once the line is selected, HyperMesh posts a message similar to:

"Gap = (.200018). Do you still wish to toggle?"

8. Click **Yes** to close the gap.



9. Click **return** to go to the main menu.

Step 9: Find and delete all duplicate surfaces.

You should still be in the **geom cleanup** panel.

1. Access the **Defeature** panel in one of the following ways:
 - From the **Pull-down** menu, choose **Geometry**, then **Defeature**
 - From the main menu, choose **Geom Page**, then **Defeature**
2. Go to the **duplicates** sub-panel.
3. Select **surfaces >> displayed**.
4. In the **cleanup tol =** field, type 0.01.
5. Click **find**.

The **message** bar displays the following message, "2 surfaces are duplicated."

6. Click **delete** to remove any duplicate surfaces.

Step 10: Observe the model again to identify any remaining free edges, or missing or duplicate surfaces.

1. Use the topology display and shaded modes to perform this task. All of the edges in the model should be displayed as green shared edges, indicating that we have a completely enclosed thin solid part.
2. Click **return** to go to the main menu.

Step 11 (Optional): Save your work

With the cleanup operations completed, save the model.

Section 2: Generating a Midsurface

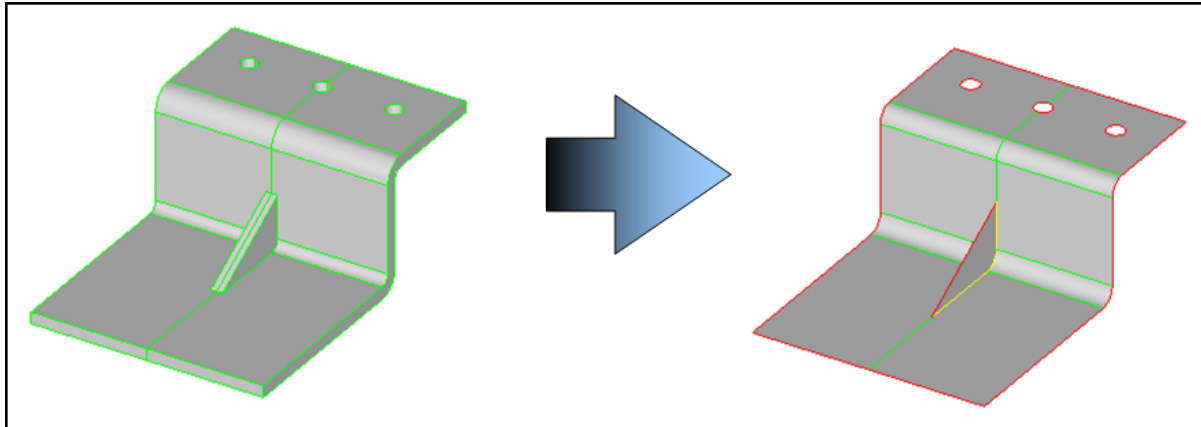
This exercise uses CAD geometry data for a thin solid clip. Because of the small thickness of the part, it is assumed that it will be modeled for FEA as shell elements. The elements will be created on the mid-plane of the part.

In this section, you will learn how to:

- Create a midsurface
- Visualize the midsurface by using shading options and transparency

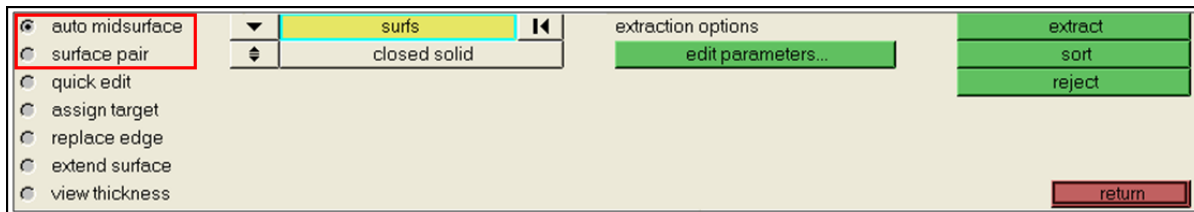
Midsurfacing: Introduction

Often the most accurate representation of a part is through the use of shell elements. These elements best represent parts that are relatively thin compared to their overall surface area and typically have a uniform thickness. Shell elements have no physical thickness representation, they are displayed as two dimensional entities whose thickness is simply a numerical value assigned to them. FE Solvers assume the shell element to lie at the middle of the thickness. As that is the case the mesh created on the surfaces needs to lie at the mid-plane of the part. CAD geometry is usually created as either a solid part or a series of faces representing a solid part. Using the midsurface tool in HyperMesh proper surfaces can be extracted that lie on the mid-plane of the part and can be meshed appropriately.



Midsurfacing: Tools

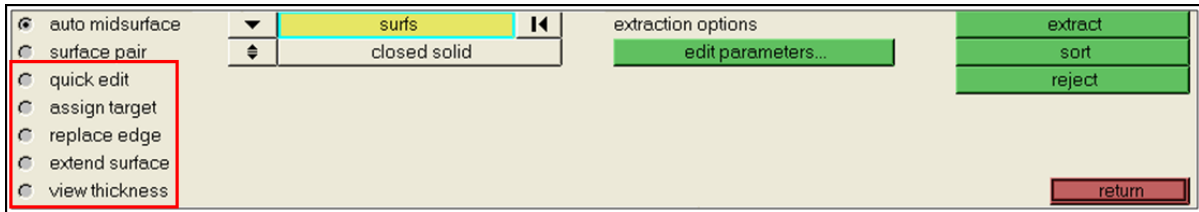
Midsurface Creation:



- **Auto Midsurface** – Automatically extracts midsurfaces from surfaces that enclose a volume or a solid geometry
 - Can sometimes work if there are missing surfaces
 - The greater number of missing surfaces, the less reliable the result
- **Surface Pair** – creates a midsurface between 2 selected surfaces

Midsurfacing: Tools cont.

Midsurface Modification



- **Quick Edit** – Repair a midsurface by correcting where the vertices of the surface were placed.
- **Assign Target** – An extension to quick edit, and functions in a similar fashion.
- **Replace Edge** – Fill in gaps and slivers by combining one surface edge with another.
 - same as in the *edge edit* panel
- **Extend Surface** – Extends two surfaces (e.g., ribs) until they intersect.
- **View Thickness** – Review of the thickness of a midsurface using white lines (probes) extending from each vertex of the surface.

Midsurfacing: Strategy




- ✓ **Obtain a closed volume of surfaces or solids**
- ✓ **For complex parts try defeaturing the defining volume**
 - This simplifies the part and can produce a better midsurface. See section 3 of this chapter for more information about defeature.
- ✓ **Generate the midsurface using the auto midsurface**
- ✓ **View the midsurface and correct any errors using modification tools.**

Exercise: Generating a Midsurface

The surfaces in the model have no connectivity errors. This could be because the file was imported without errors or because the errors were corrected using HyperMesh. In this case, errors in the topology were repaired in the previous exercise (missing surfaces are re-created, duplicate surfaces are deleted, gaps are closed, etc.). You can use either the file created from the previous exercise, or open the new, `clip_midsurface.hm`, file. Either way, the geometry is at the point where you can use the **midsurface** panel to generate a midsurface.

Step 1: Retrieve and view the model file.

This exercise uses the model, `clip_midsurface.hm`.

1. On the **visualization** toolbar, change the visualization to **Shaded Geometry & Surface Edges** ().
2. On the visualization toolbar, set the **Geometry Color** drop-down menu to **Mixed** ().
3. To change the model back to wireframe click **wireframe** (.

Notice how the surfaces represent a solid part, these visualization techniques will be necessary for viewing the newly created midsurface.

Step 2: Generate a midsurface from **midsurface** panel.

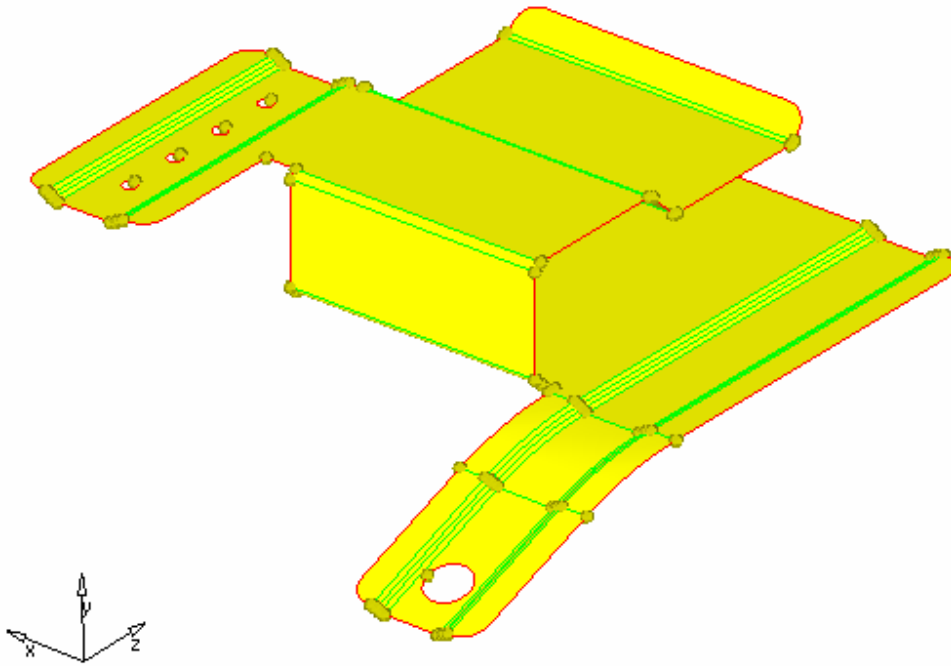
1. Access the **midsurface** panel by doing one of the following:
 - From the **Pull-down** menu, select **Geometry**, then **Midsurface**
 - From the main menu, select the **Geom** page, then select **midsurface**
2. Go to the **auto midsurface** panel.
3. Verify that the **closed solid** option and the yellow **surfs** selector are active.
4. Select one surface from the graphics area.
5. Click **extract** to start the midsurface generation.

The midsurface is created, and the surfaces are organized into a new **Middle Surface** component. Note that when a midsurface is created, transparency is turned on for all the other components in the model except the new **Middle Surface** component. The next step covers how to control surface transparency.


Step 3: Review the part's midsurface.

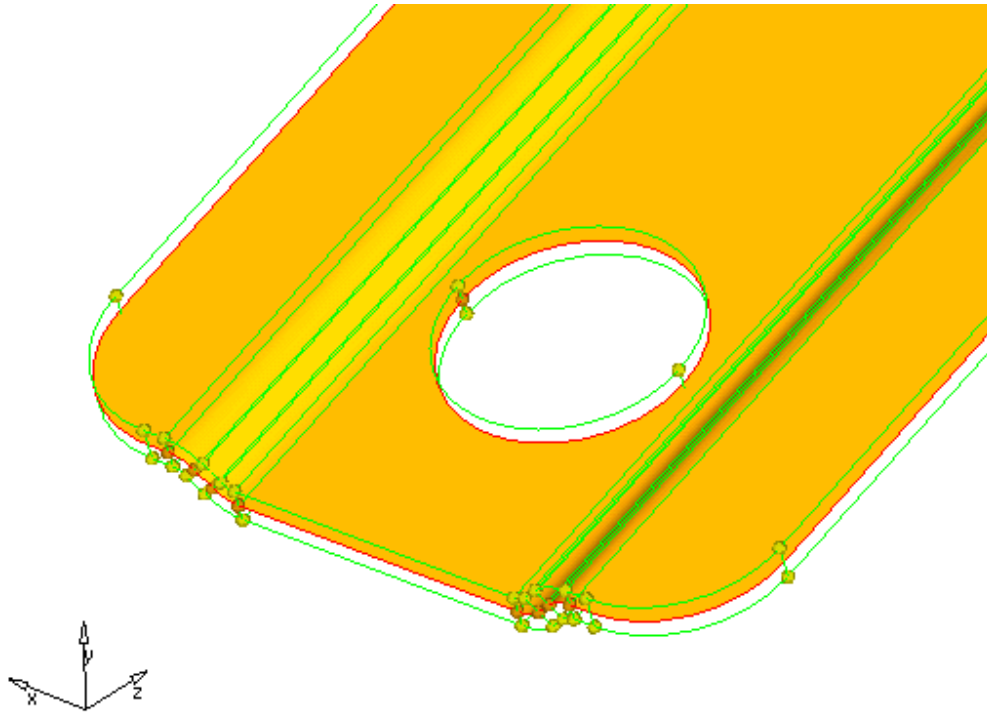
1. From the model browser, turn off the display of geometry in the component named **lv10** to display only the **Middle Surface** component.

The midsurfaces generated for the solid sections of the model using the solid sub-panel are shown in the following image.



Midsurface generated from a volume of surfaces

2. Turn the geometry for **lvl10** component back on from the **model browser**.
3. On the **visualization** toolbar, access the **transparency** panel ().
4. With the **comps** selector active, select a line or surface of the **lvl10** component in the graphics region.
 You may need to zoom in on the model to select a valid entity.
 Since the entity selector is set to comps, selecting a line or surface in the component selects the entire component.
5. In the panel, click the arrow button on the right under **transparency** several times.
 You should see the surfaces in the **lvl10** component becoming more and more transparent.
 You can also drag the slider back and forth to control the level of transparency.
6. Rotate, zoom, and pan to visualize the midsurface.



Close-up of the midsurface with the **lv10** component set to full transparency

7. Click **wireframe** () on the **visualization** toolbar.

Step 4 (Optional): Save your work.

Now that the midsurface has been created, it is a good time to save the model.

Summary

We now have surfaces created on the mid-plane of the part. These surfaces can now be meshed or further modifications can be made to their topology, depending on the requirements of the analysis.

Section 3: Simplifying Geometry

This section looks at changing the shape of a part in order to simplify the geometry. Certain details of the shape, such as small holes or blends, may simply not be necessary for the analysis being performed. When these details are removed, the analysis can run more efficiently. Additionally, mesh quality is often improved as well. Changing the geometry to match the desired shape can also allow a mesh to be created more quickly.

In this section, you will learn:

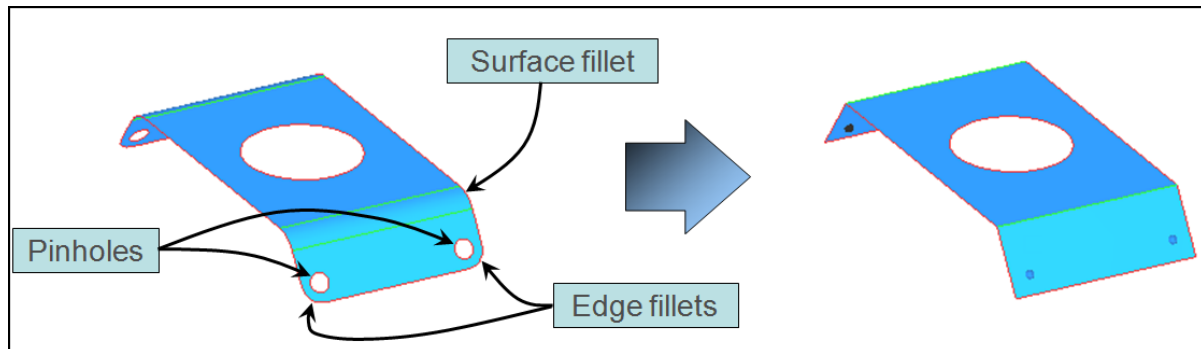
- Mesh the clip, review the mesh quality, and determine the features to be simplified
- Remove surface fillets
- Remove edge fillets
- Remove pinholes

Defeaturing

There are many features on a part that are not critical to the structure of the part and have little or no effect on the analysis. These features can include

- **Lightening Holes** – For part weight reduction
- **Edge Fillets** – For reduction of sharp corners allowing safer part handling
- **Surface Fillets** – To meet manufacturing requirements

These features often are process driven and are not function critical. While our goal is to mesh a part that as closely as possible accurately represents the geometry, these features often degrade the quality of the mesh. As such they can be defeated out of the design allowing for a substantially improved mesh with little impact on the results.

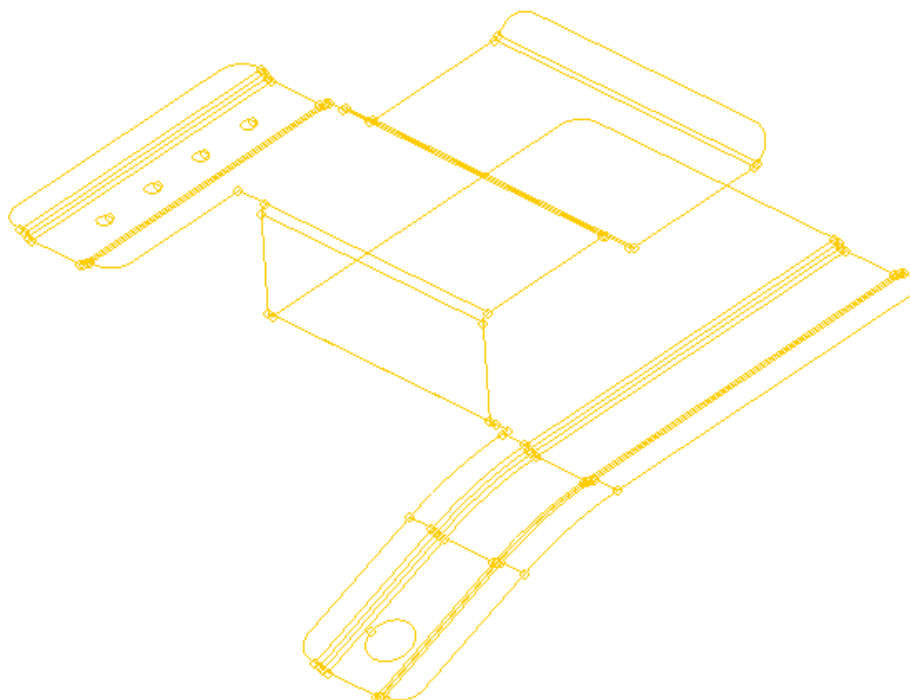


Defeaturing Tools

- **Pinholes**
 - Searches for holes within a surface. Fills them in and leaves a fixed point at their previous center.
- **Surf Fillets**
 - Searched for surfaces that act as a fillet between other surfaces and tangentially extends them to achieve a sharp corner.
- **Edge Fillets**
 - Searches for rounded edge corner and squares them off.
- **Duplicates**
 - Finds and deletes duplicate surfaces.
- **Symmetry**
 - Identifies part symmetry and deletes or organizes the results.

Exercise: Removing Details from a Midsurface Model

This exercise uses the model file, `clip_defeature.hm`. The model file has geometry that has been midsurfaced. (Surfaces have been created on the mid-plane of the part.) The model will be meshed using an element size of 2.5. You can assume a simple structural analysis will be run on the part, and thus does not require much detail. Based on this, there are features, which are not necessary and can be removed.



Step 1: Open and view the model file, `clip_defeature.hm`.


1. From the **File** pull-down menu on the toolbar, click **Open**.
2. Select the `clip_defeature.hm` file and click **Open**.

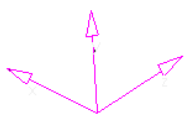
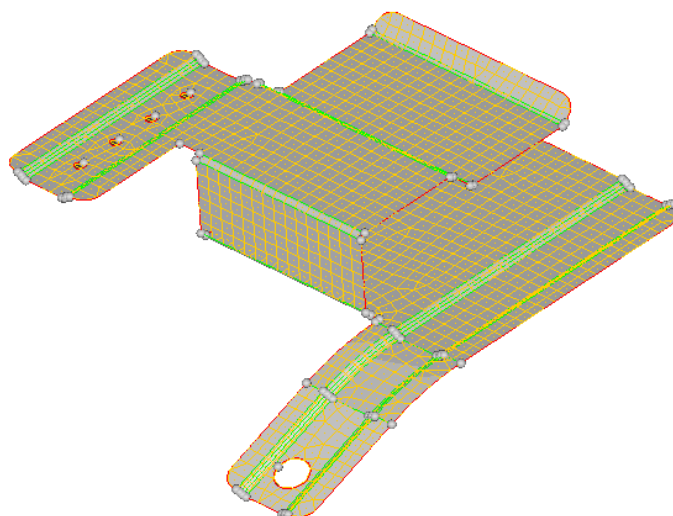
Step 2: To easily work with the midsurface, turn off the display of the **lvl10** component.

If the **lvl10** component is displayed, it needs to be turned off so that you can easily work on the midsurface geometry. (It may be on if you used the model you had open for the previous section.)

1. Navigate to the **model browser**.
2. Verify that the **Elements/Geometry** selector is set to **Geometry**.
3. If the geometry display for the the component **lvl10** is selected as active, click the icon for the **lvl10** component to turn off the display of the geometry in that component.

Step 3 (Optional): Mesh the clip to view mesh quality before defeaturing.

1. Click **Shaded Geometry** and **Surface Edges** (.
2. Access the **automesh** panel:
 - From the **Pull-down** menu, choose **Mesh**, then **Create**, then **2D Automesh**
3. Set the selector type to **surfs**.
4. Go to the **size and bias** sub-panel.
5. In the **element size =** field, type 2 . 5.
6. For **mesh type:**, select **mixed**.
7. Switch the **meshing mode** from **interactive** to **automatic**.
8. Verify that the **elems to surf comp** toggle is set.
9. Select **surfs >> displayed** to select all displayed surfaces.
10. Click **mesh** to generate the mesh preview.



Initial mesh on the clip model

11. Click **return** to go to the main menu.

Step 4 (Optional): Review the quality of the mesh.

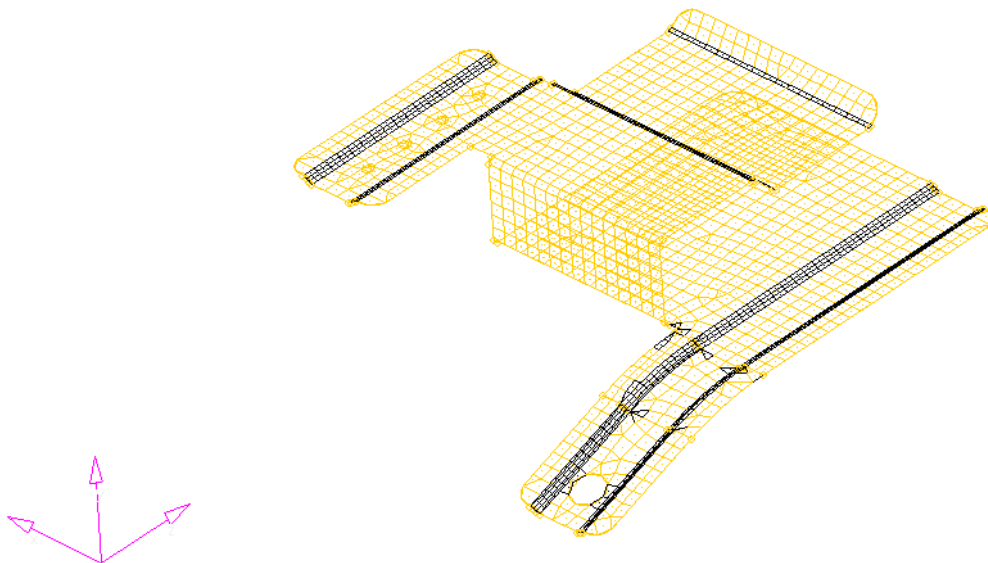
Take a minute to look over the mesh that was created. Note the areas that have irregular, poor quality mesh. Also, use the check elems panel to evaluate the minimum length check of the elements.

Take a minute to rotate, zoom, and pan the model to review the mesh that was created. Note the locations where the mesh was not created in neat rows and columns of quads.


1. Access the **check elems** panel by doing one of the following:

- From the **Pull-down** menu, choose **Mesh**, then **Check**, then **Elements**, then **Check Elements**
 - From the main menu, select the **Tool** page, then select **check elems**
2. Access the **2-d** sub-panel.
 3. In the **length** field, type 1.
 4. Click **length** to evaluate the minimum length.

Many of the elements failing the length test are located around the fillets of this model.



Elements failing the length check

Note: You may need to change the geometry display to wireframe for better visualization of element quality. For this, click **Wireframe Geometry** ().

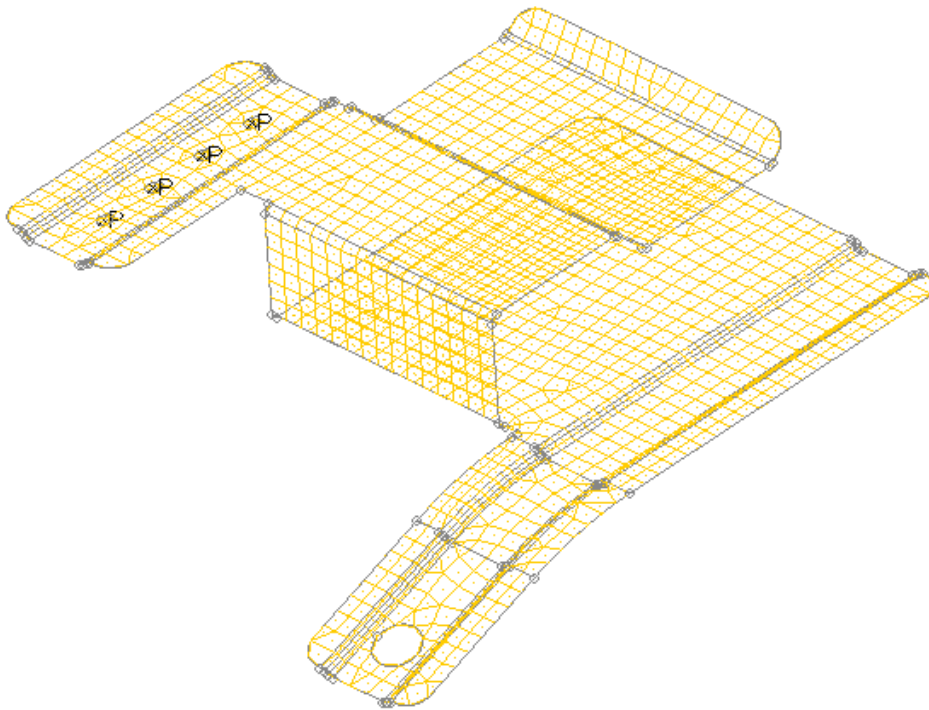
5. Click **return**.
6. Use the **model browser** to turn off the display of the elements in the **lv10** component.

Step 5: Remove the four small pinholes.

Pinholes are closed free edge loops within a surface. Pinholes do not need to be circular.

1. Access the **defeature** panel by doing one of the following:
 - From the **Pull-down** menu, select **Geometry**, then **Defeature**
 - From the main menu, click **Geom Page**, then click **Defeature**
2. Go to the **pinholes** sub-panel.
3. In the **diameter** field, type 3.0.
4. Select **surfs >> all**.
5. Click **find** to identify the pinholes having a diameter of 3 or less.

Notice the xP symbol at the center of the four circular holes. These symbols are highlighted in white, indicating they are pinholes identified by HyperMesh for removal.




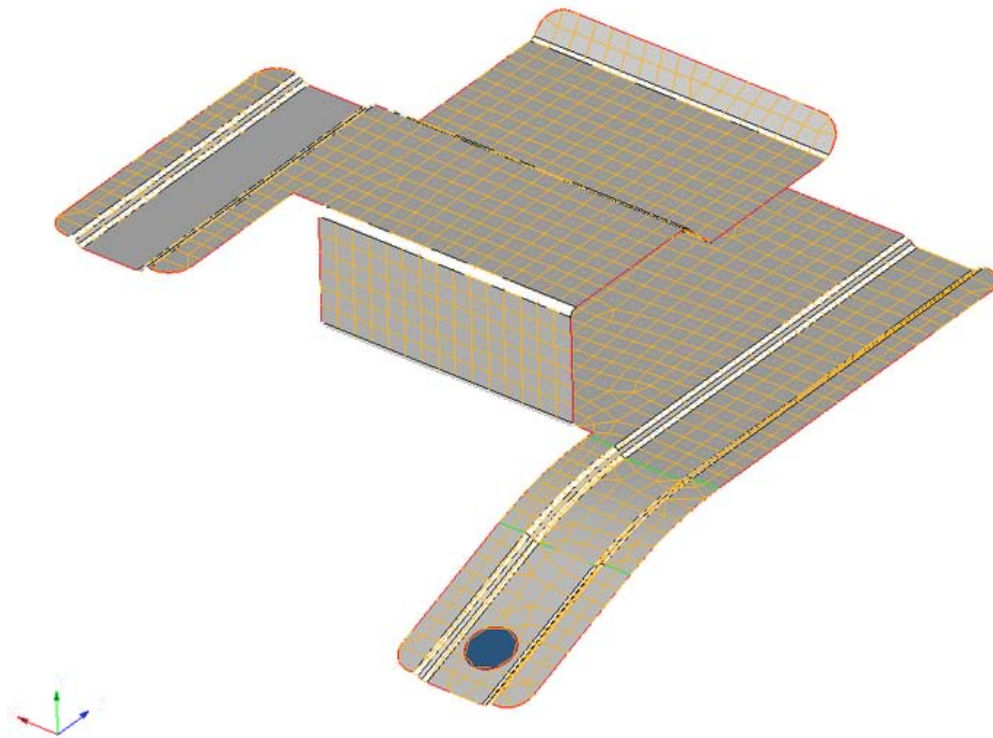
Pinholes identified using a 3 mm diameter

6. Click **delete** to remove the selected pinholes in the model.

The selected pinholes are removed and replaced by fixed points located at the center of the original pinholes.

Step 6: Remove all surface fillets in the clip.

1. Go to the **defeature** panel.
2. Go to the **surf fillets** sub-panel.
3. If the surfaces are not shaded, click **Shaded Geometry and Surface Edges** (.
4. To find fillets in displayed surfaces, select **surfs >> displayed**.
5. In the **min radius** field, type 2 . 0.
6. Click **find** to identify all the surface fillets with a radius of 2 or greater.



Surface fillets identified for removal

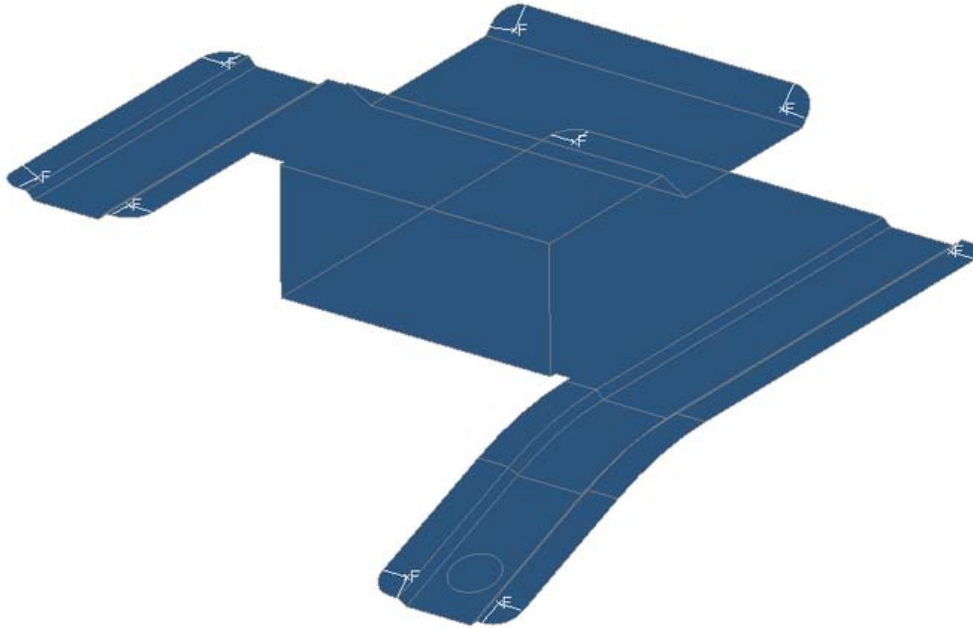
7. Click **remove**.

Step 7: Automatically identify and remove rounded corners of surfaces.

You should still be in the **defeature** panel.

1. Go to the **edge fillets** sub-panel.
2. Select **surfs >> displayed**.
3. In the **min radius** field, type 1 . 0.
4. Set the bottom switch to **all** to find all the fillets.
5. Click **find**.

The edge fillets that meet the filter criteria are identified on the screen with an F symbol and radial lines marking the fillet beginning and end.



Edge fillets identified for removal

6. Notice that the selector moves down to the **fillets entity** selector.
7. Right-click on one of the **F** fillet markers on the screen to deselect the fillet.
8. Click **remove** to delete the selected edge fillets.

All the fillets are removed.

Step 8: Mesh the defeatured geometry and view the quality.

1. Go to the **automesh** panel.
2. Select **surfs>>displayed**.
3. Click **mesh**.

Observe the mesh and you will notice that the mesh is created in rows of quads.

Step 9 (Optional): Save your work.

Now that the model has been simplified, it is a good time to save the model.

Summary

The model is now represented in a much simpler form that suits the analysis that will be performed. Holes, surface fillets, and edge fillets were removed that were considered too small to be captured by the desired element size of 2.5.

Section 4: Refining Topology to Achieve a Quality Mesh

Once a set of surfaces has been obtained that accurately reflects the intended shape for the analysis, further geometry cleanup may still be necessary. Topological details of the geometry may affect the quality of the mesh created from the surfaces. These details may not reflect any major feature of the part's shape, and can be removed without concern. When modifying the topology affects the shape of the surfaces, a compromise must be made between the part shape and the element quality necessary for the analysis. Other times, adding topological features that do not change the shape of the part may actually help create a better quality mesh.

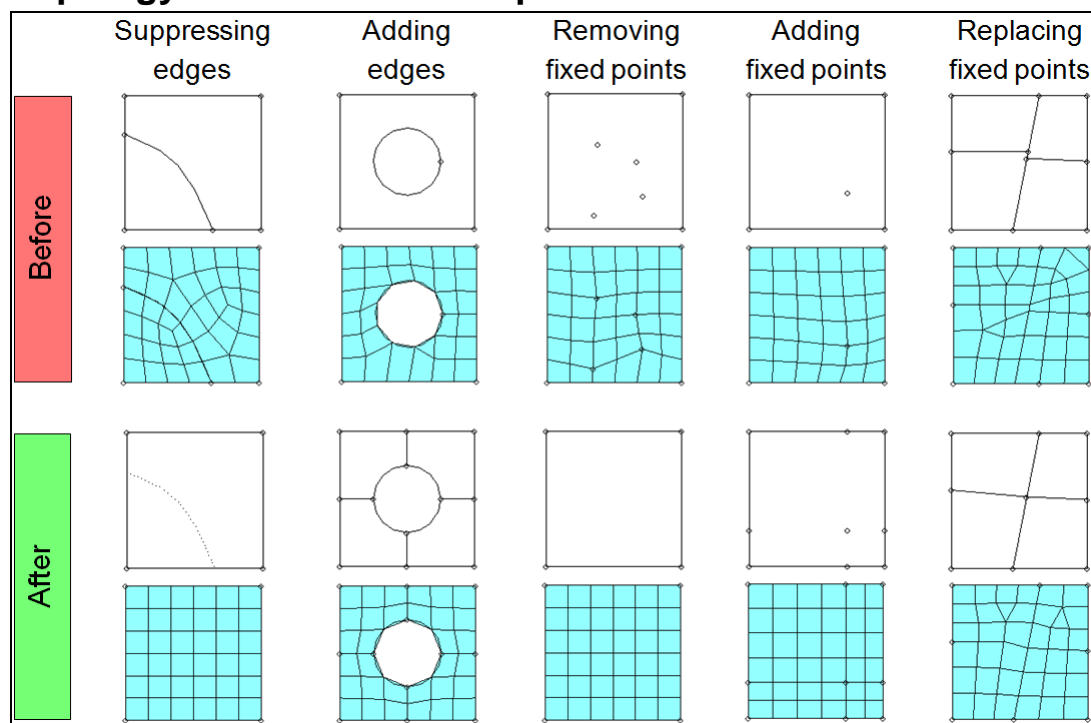
In this section, you will learn how to:

- Mesh the part to determine poor element quality
- Suppress small edges
- Split surfaces
- Remove interior fixed points
- Replace closely placed fixed points
- Create final mesh

Topology Refinement: Introduction

- “Topology refinement” is modifying topology in order to obtain a quality mesh
 - Unlike defeaturing, this generally does not change the shape of the part.
- CAD geometry has topology details that interfere with mesh quality.
 - Edges are created where ever there is a change in surface curvature.
 - Even smooth areas can be split into several faces.
 - The automesher will be forced to place nodes along the edges and fixed points.
 - This may cause small element lengths, angles, aspect ratios, etc. depending on the shape of the model.
- CAD geometry can also have surfaces with a complex outline.
- Highly complex shapes can make it hard to get a quality mesh.
 - Adding in edges splits the surfaces into smaller and simpler regions which can be meshed easier.
 - Fixed points can be added in to force a node to be placed at that location, giving more control.

Topology Refinement: Examples



Topology Refinement: Tools

- **Edge edit**
 - **Toggle** – Select a shared edge to suppress it.
 - **(Un)Suppress** – Select multiple edges to suppress all of them at once.
- **Point edit**
 - **Add** – Add fixed points to help control mesh pattern.

- Especially helpful along edges to control node seeding.
- **Suppress** – Removes unwanted fixed points.
- **Replace** – Combines 2 fixed points at a single location.
- **Project** – Projects fixed points onto a nearby edge.
 - Useful for aligning mesh between 2 edges.
- **Surface edit – All functions add edges by cutting surfaces**
 - Adding edges can be useful for controlling mesh patterns in large areas.
 - **Trim with nodes** – Uses node locations to cut surfaces.
 - **Trim with lines** – Uses lines to cut surfaces.
 - **Trim with surfs/plane** – uses other surfaces or a defined plane to cut surfaces.
- **Quick edit**
 - **Split surf-node:** Adds an edge to or divides a surface by cutting in a straight line between 2 selected nodes.
 - **Split surf-line:** Adds an edge to or divides a surface by cutting a straight line between a node and perpendicular to the selected line.
 - **Washer split** – Adds a circular edge around a hole in a surface Mostly used for creating all quad mesh around a hole.
 - **Toggle** – Same as edge edit panel.
 - **Remove Point** – Deletes a selected fixed point.
 - **Replace point** – Same as point edit panel.
 - **Add/remove point : point** – Creates a new fixed point at the selected locations.
 - **Add point: line** – Creates a user specified number of fixed points along the selected edge.
 - **Project point** – Same as edge edit panel.

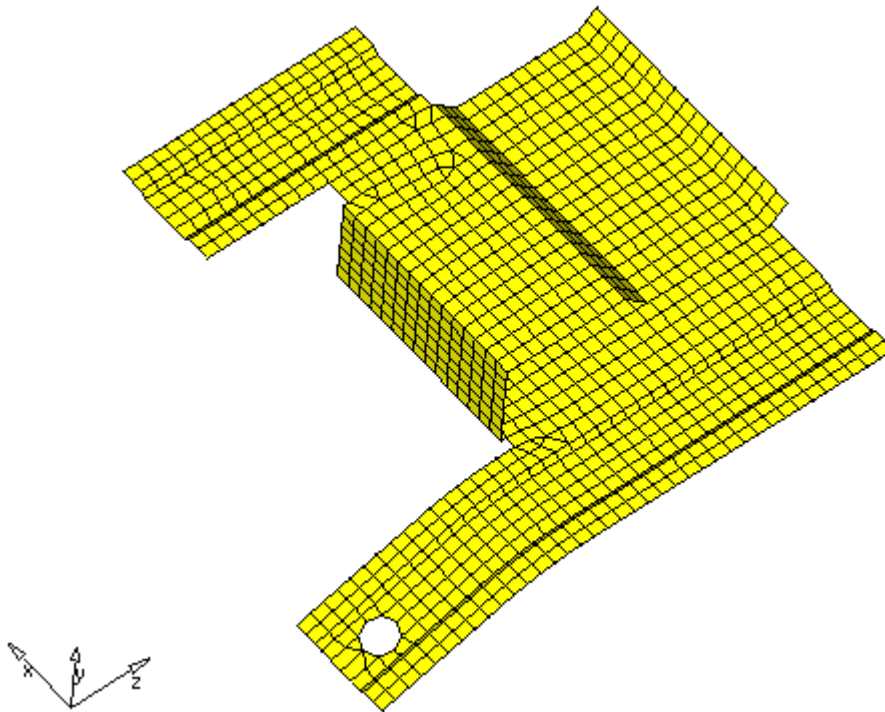
Exercise: Refining Topology to Achieve a Quality Mesh

Step 1: Open the model file, `clip_refine.hm`.

The model for this exercise is `clip_refine.hm`. Take a few moments to observe the model using the different visual options available in HyperMesh (rotation, zooming, etc.).

Step 2: Create a preliminary mesh.

1. Access the **automesh** panel by doing one of the following:
 - From the **Pull-down** menu, select **Create**, then **2D Automesh**
 - From the main menu, select the **2D** page, then click **automesh**
2. Set the selector type to **surfs**.
3. Go to the **size and bias** sub-panel.
4. In the **element size =** field, type 2 . 5.
5. For **mesh type:**, select **mixed**.
6. Switch the meshing mode from **interactive** to **automatic**.
7. Select **surfs >> displayed**.
8. Click **mesh** to mesh the surfaces.



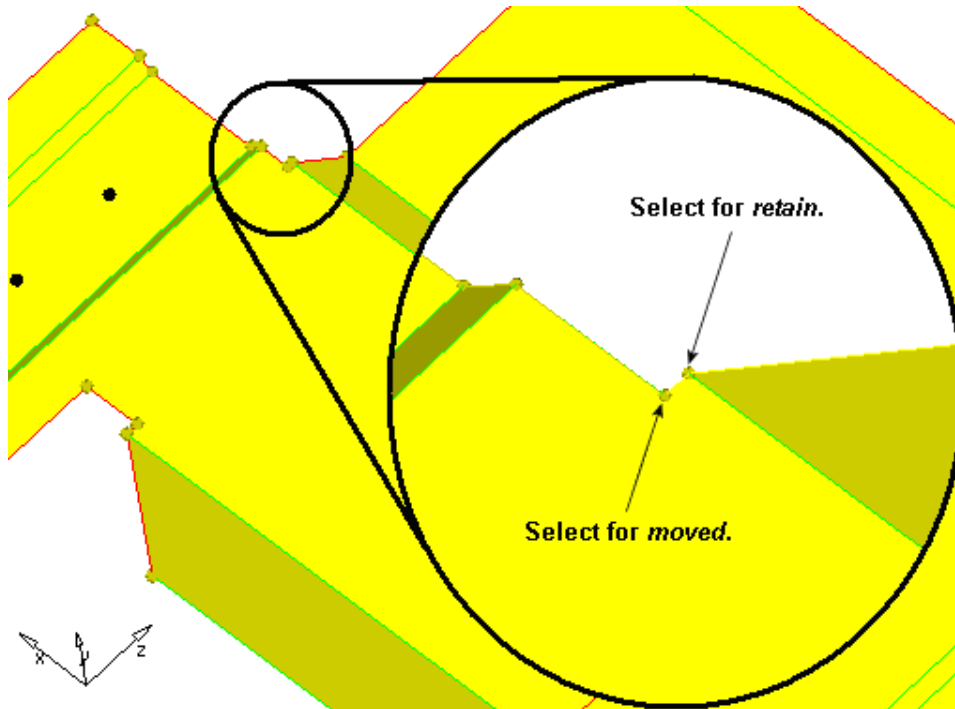
Initial mesh on the defeatured clip model

Step 3: Review the mesh quality.

1. Take a minute to rotate, zoom, and pan the model to review the mesh that was created. Note the locations where the mesh was not created in rows and columns of quads.
2. Access the **Check Elements** panel by doing one of the following:
 - From the **Pull-down** menu, select **Mesh**, then **Check**, then **Elements**, then **Check Elements**
 - From the main menu, access the **Tool** page, and then click **check elems**
3. Go to the **2-d** sub-panel.
4. In the **length** field, type 1.
5. Click the **length** button to evaluate the minimum length.
6. Note the elements that failed the check. The topology will be edited to correct some of these, and the others will be left as is.
7. Click **return** to return to the main menu.
8. Use the **model browser** to turn off the display of the elements in the **lv10** component.

Step 4: Remove short edges by combining fixed points.

1. Access the **point edit** panel by doing one of the following:
 - From the **Pull-down** menu, select **Geometry**, then **Edit**, then **Points**.
 - From the main menu, select the **Geom** page, and then click **point edit**
2. Go to the **replace** sub-panel.
3. Verify that the active selector is set to **moved points**.
4. Select the lower fixed point as indicated in the following image.
5. Once the point is selected, activate the **retained point** button.
6. Select the upper fixed point as indicated in the following image.
7. Once the second point is selected, click **replace**.



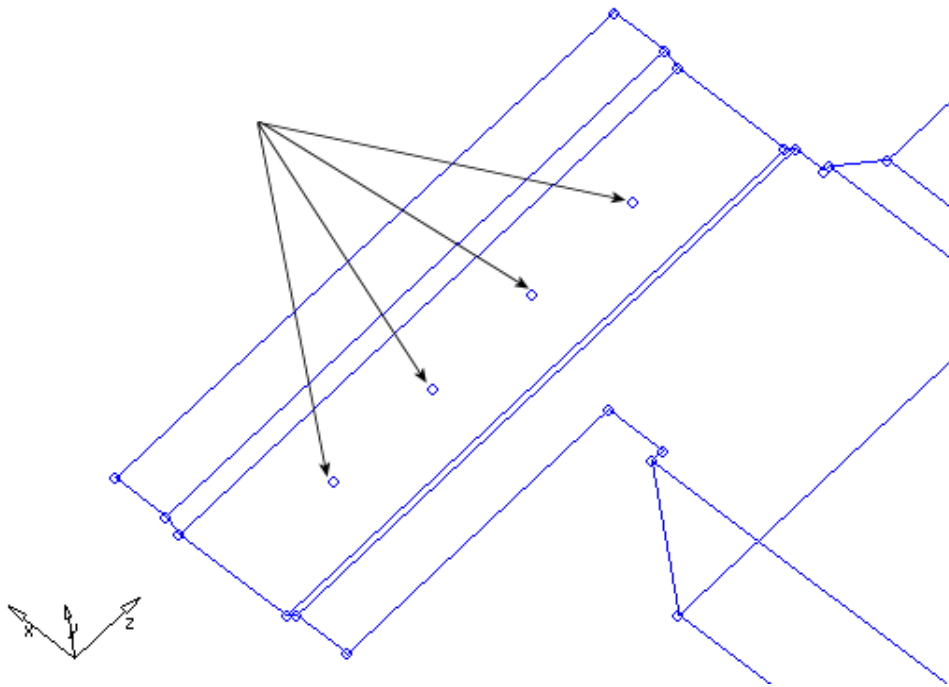
Selecting fixed points to be combined

Step 5: Remove the fixed points interior to all surfaces.

1. You should still be in the **points** panel.
2. Go to the **suppress** sub-panel and choose **at cursor** option.
3. Select the four fixed points as shown in the following image.

Each fixed point will be deleted as you select it.

These fixed points are left over from a defeaturing operation where small holes were removed. They could remain without greatly sacrificing the element quality, given the element size used for the mesh.



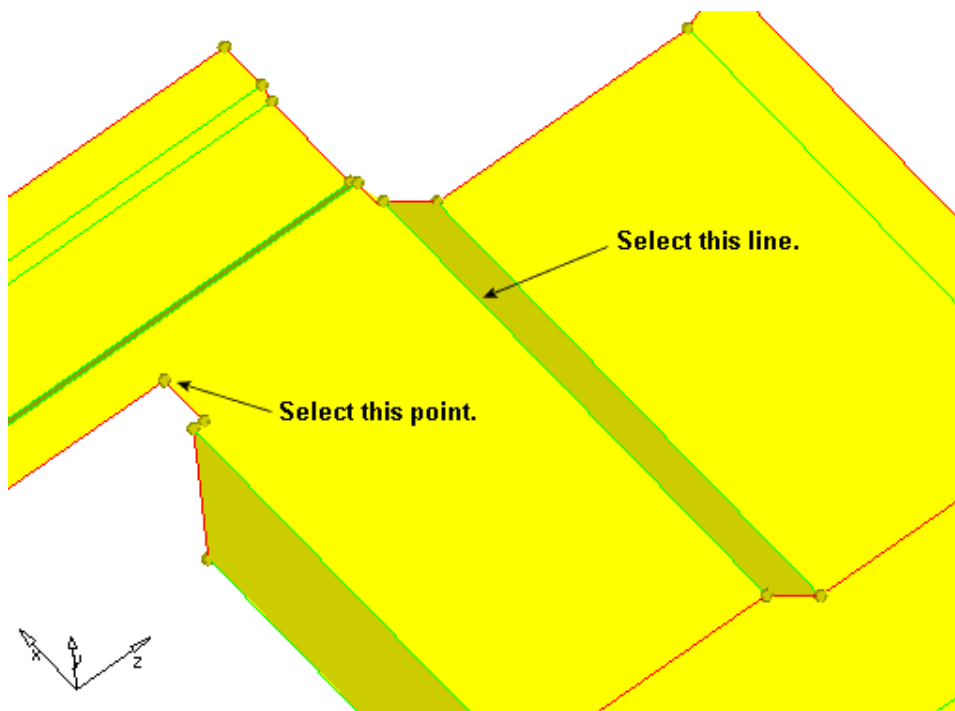
Fixed points to be removed

4. Click **return** to go to the main menu.

Step 6: Add edges to the surfaces to control the mesh pattern.

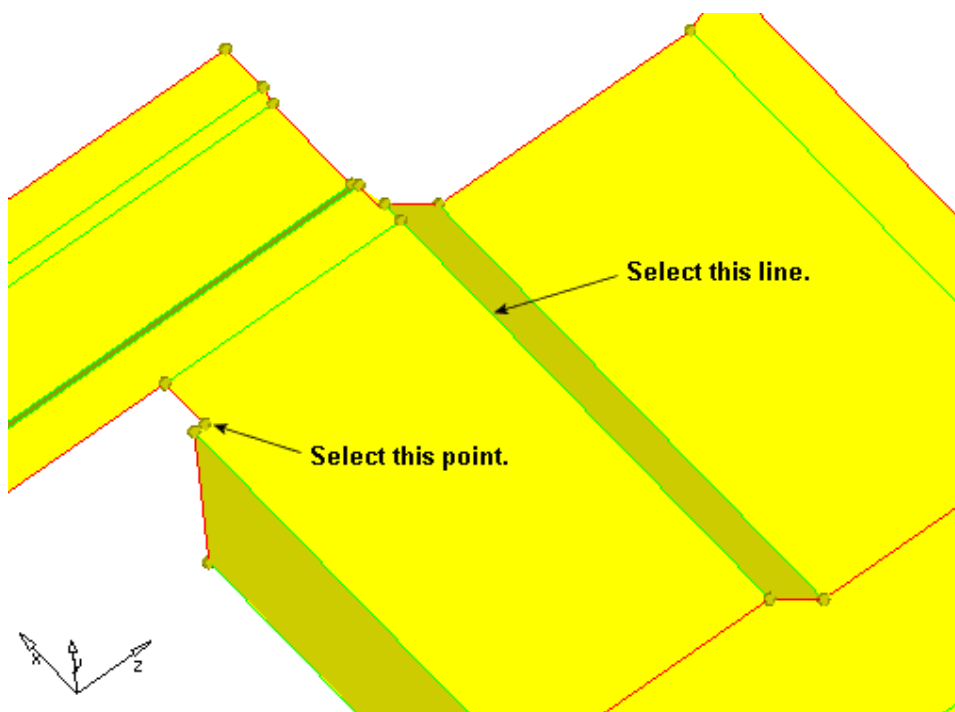
1. Access the **surface edit** panel by doing one of the following:
 - From the **Pull-down** menu, select **Geometry**, then **Edit**, then **Surfaces**.
 - From the main menu, select the **Geom** page, then select **surface edit**.
2. Go to the **trim with nodes** sub-panel.
3. Under **node normal to edge**, set the active selector to **node**.
4. Zoom into the area indicated below and select the indicated fixed point.
5. With the active selector now on **lines**, select the line shown in the following image.

Once both the point and line are selected, an edge will be created from the location of the fixed point perpendicular to the line.



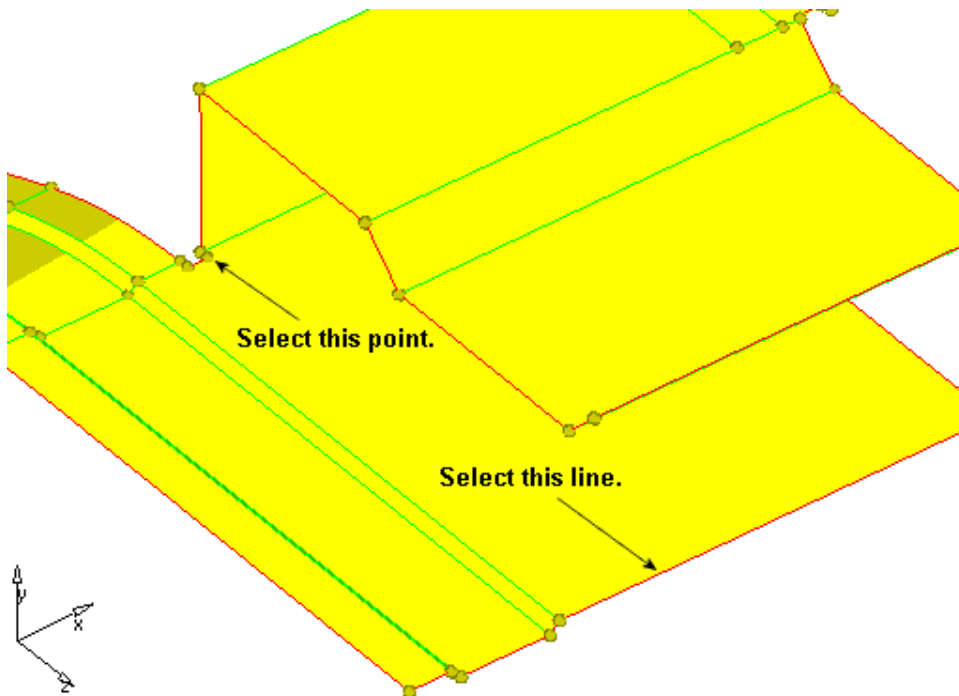
Select fixed point and line to split the surface.

6. Repeat #3, #4 and #5 for the following point and line.



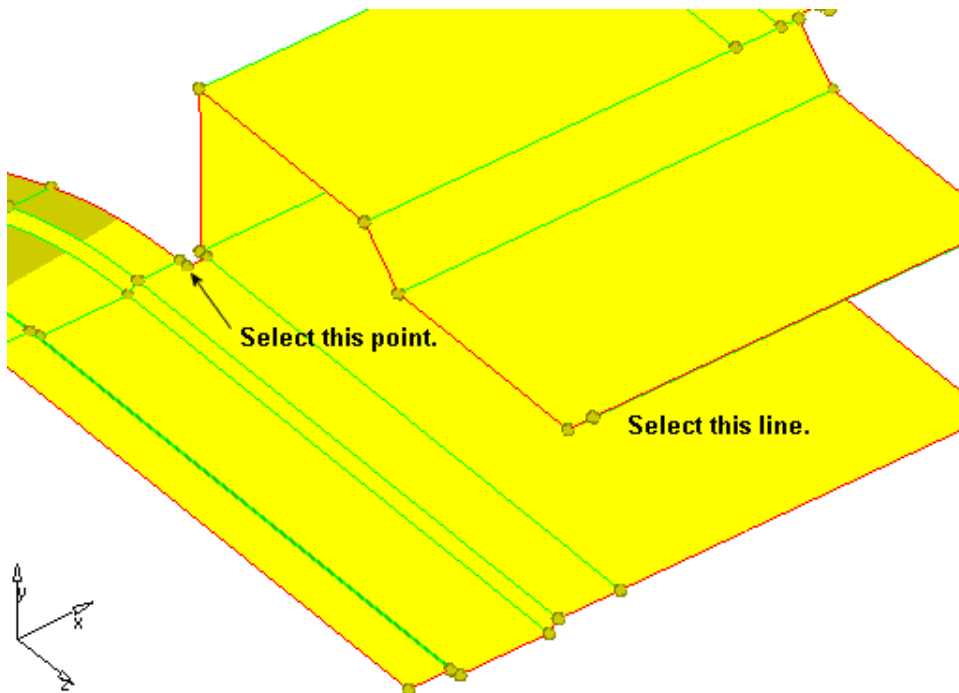
Select fixed point and line to split the surface.

7. Repeat #3, #4 and #5 for the following point and line.



Select fixed point and line to split the surface.

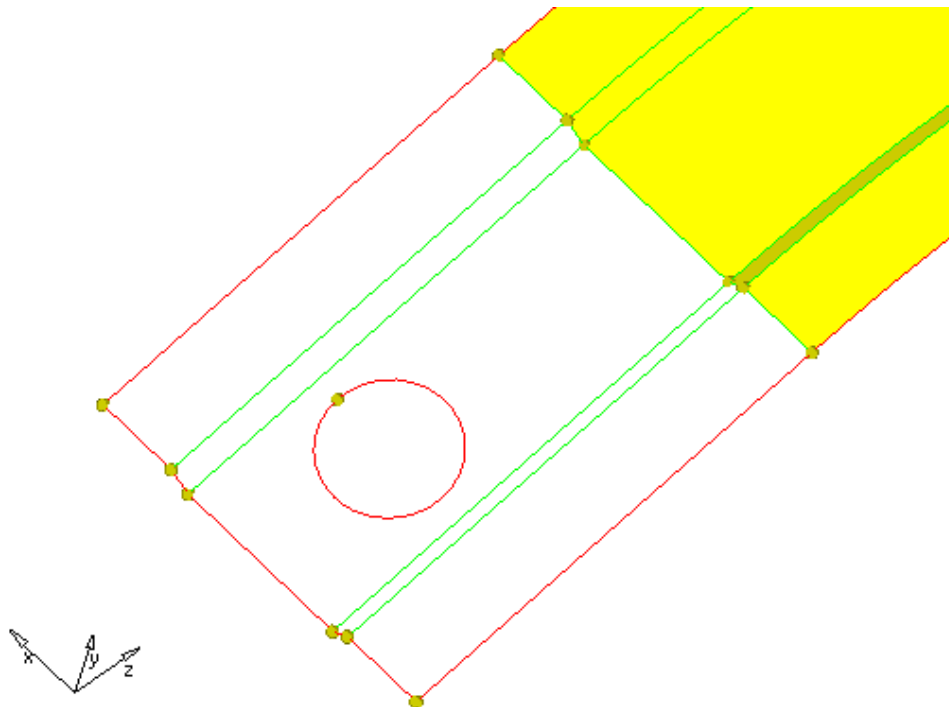
8. Repeat #3, #4 and #5 for the following point and line.



Select fixed point and line to split the surface


Step 7: Add edges to the surfaces to control the mesh pattern.

1. Access the **surface edit** panel.
2. Go to the **trim with surfs/planes** sub-panel.
3. In the **with plane** column, set the active selector to **surfs**.
4. Select the surfaces indicated in the following image.



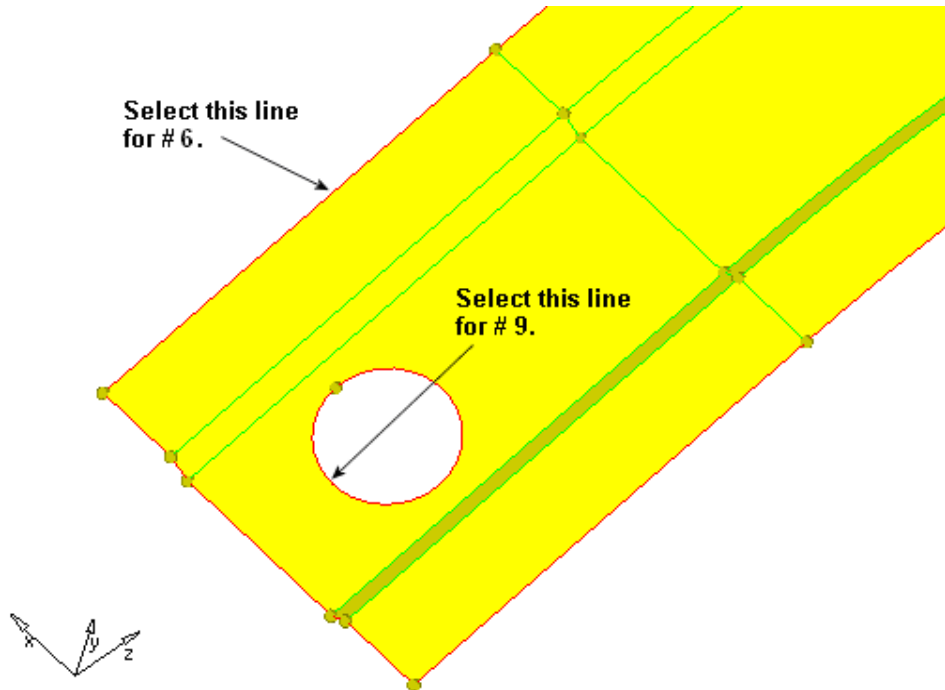
Surfaces to be selected for splitting

5. If necessary, toggle the **direction** selector to **N1**, **N2**, and **N3**. Click **N1** to make the selector active.
6. Press and hold your left mouse button, and then move it over the edge indicated in the following figure.

Once over the line, the cursor will change to a square with a dot in the center, . Release your mouse button. Click two points anywhere along the edge. Do not click a third.

Nodes will be placed on the line for **N1** and **N2**.

7. Press F4 on the keyboard to enter the **distance** panel.
8. Go to the **three nodes** sub-panel.
9. As in step #6, press and hold your left mouse button, and then move it over the edge of the hole, as indicated in the following image. Once over the line, the cursor will change to a square with a dot in the center. Release your mouse button.



Select fixed point and line to split the surface

10. Click three points anywhere along the edge.

Temporary nodes will be placed on the line representing N1, N2 and N3.

Note that the technique used to create nodes to select where none existed before can be used in any place where nodes need to be selected but don't exist in the model. You can create nodes in this manner on lines, surfaces and elements. For more details, see the HyperMesh online help. Pick the index and type, Picking Nodes on Geometry or Elements.

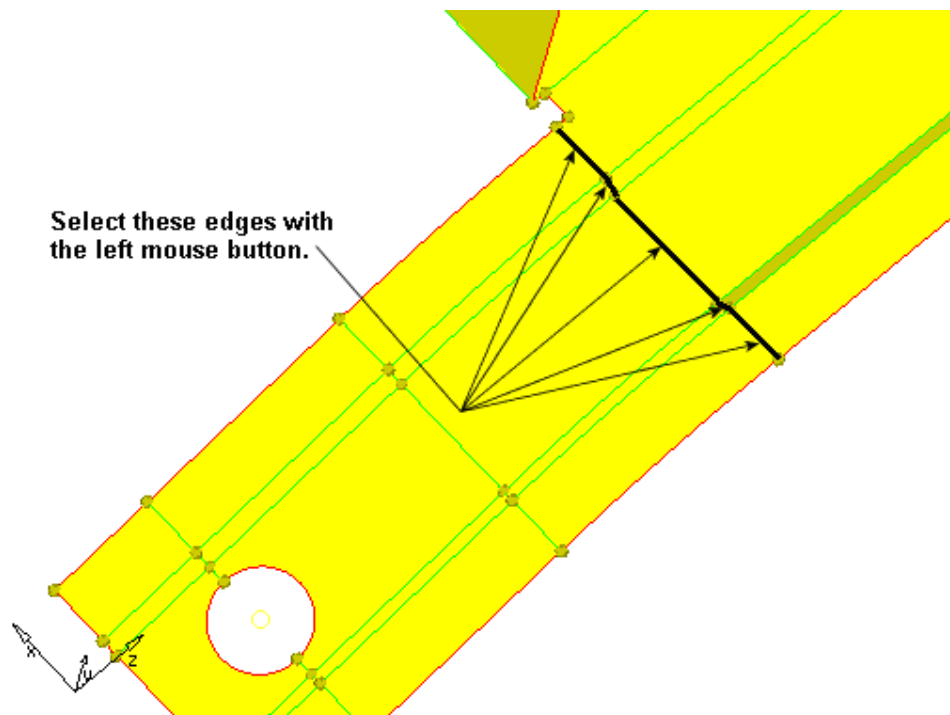
11. Click **circle center** to create a node at the center of the hole.
12. Click **return** to go to the **surface edit** panel.
13. Click **B** to make it the active selector.
14. Select the node that was just created at the center of the hole.
15. Click **trim**.
16. Click **return** to go to the main menu.

The surfaces are trimmed through the center of the hole.

Step 8: Suppress shared edges causing a small edge.

1. Go to the **edge edit** panel.
2. Go to the **(un)suppress** sub-panel.
3. Select each of the lines in the image below using your left mouse button and click **suppress**.

Each line will become suppressed (blue) as you click suppress.



Surface edges to suppress by toggling

Step 9: Remesh the part.

Remesh the surfaces of the part, using the automatic mode, a size of 2.5, and the mixed mesh type.

1. Use the **model browser** to display the elements in the **lvl10** component.
2. Access the **automesh** panel.
3. Verify that **elem size** = is set to 2.5 and the **mesh type** is set to **mixed**.
4. Select **surfs >> displayed** to select all displayed surfaces.
5. Click **mesh**.

Step 10: Review the mesh quality.

1. Take a minute to rotate, zoom, and pan the model to review the mesh that was created. Note that the mesh now consists completely of rows and columns of quads.
2. Enter the **Check Elements** panel.
3. Go to the **2-d** sub-panel
4. In the **length** field, type 1.
5. Click the **length** button to evaluate the minimum length.

Note the elements that failed the check. There are only two elements, and these fail the check because of the shape of the part. However, they are not too small compared to the global element size, so you can leave them as they are.

6. Access the **automesh** panel.
7. Go to the **QI optimize** sub-panel.
8. Verify that **elem size =** is set to 2.5 and the mesh type is set to **mixed**.
9. Click **edit criteria**.
10. In the **Target element size** field, type 2.500.
11. Click **Apply** and **OK**.
12. Select **surfs >> displayed** to select all displayed surfaces.
13. Click **mesh**.

Note that the old mesh is replaced by the new mesh.

14. If there is a message saying, "There is a conflict between the user requested element size and quality criteria ideal element size," click the button, Recompute quality criteria using size of 2.5.
15. Access the **qualityindex** panel by doing one of the following:
 - From the **Pull-down** menu, select **Mesh**, then **Check**, then **Elements**, then **Quality Index**
 - From the main menu, select the **2D** page, then select **qualityindex**
16. Go to **pg1** and verify that the **comp. QI** is 0.01. (If your value is higher than this number than you can try to recalculate by going back to Step 6 and performing the steps again, selecting the surfaces before entering the 2.5 element size.)

This low value indicates that the mesh is good quality. The higher the number, the lower the mesh quality.

Step 11 (Optional): Save your work

The part is now meshed and ready to be set up for an analysis. Save the model, if desired.

Chapter 3

Shell Meshing

Section 1: AutoMeshing

The optimal starting point for creating a shell mesh for a part is to have surface geometry defining the part. The most efficient method for creating a mesh representing the part includes using the **automesh** panel and creating a mesh directly on the part's surfaces.

The **automesh** panel is a key meshing tool in HyperMesh. Its meshing module allows you to specify and control element size, density, type, and node spacing, and also perform quality checks before accepting the final mesh.

A part can be meshed all at once or in portions. To mesh a part all at once, it may be advantageous to first perform geometry cleanup of the surfaces, which can be done in HyperMesh.

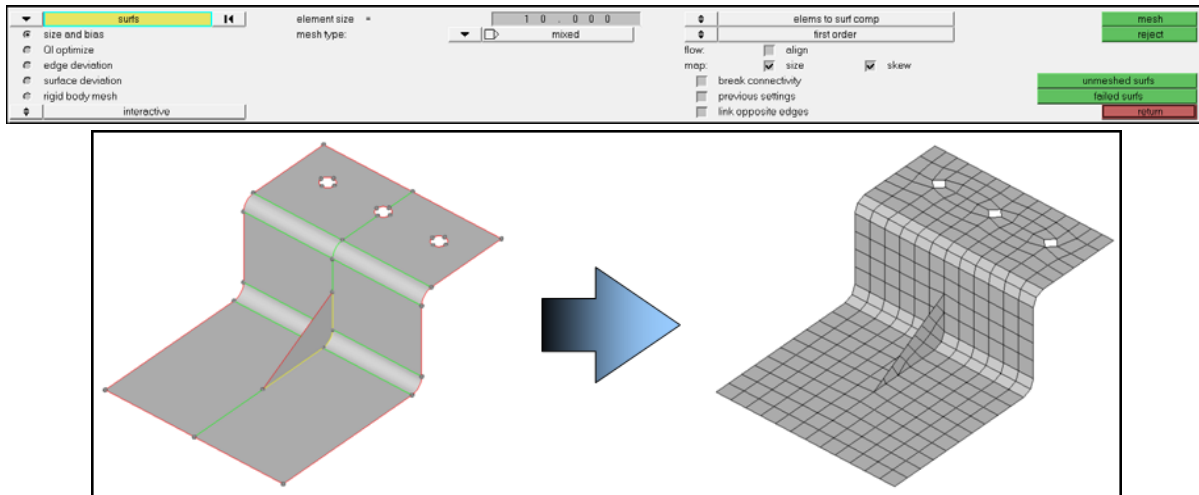
The optimal starting point for creating a shell mesh for a part is to have geometry surfaces defining the part. The most efficient method for creating a mesh representing the part includes using the **automesh** panel and creating a mesh directly on the part's surfaces.

The purpose for this section is to help you become proficient with using the **automesh** panel's meshing module. In this section, you will learn how to:

- Mesh all the surfaces at once specifying different element sizes and element types.
- Practice changing the element density along surface edges.
- Practice checking element quality and changing the mesh pattern by changing the mesh algorithm.
- Preview the mesh on all the unmeshed surfaces.
- Practice changing the element type and node spacing (biasing) along surface edges.
- Re-mesh surfaces.

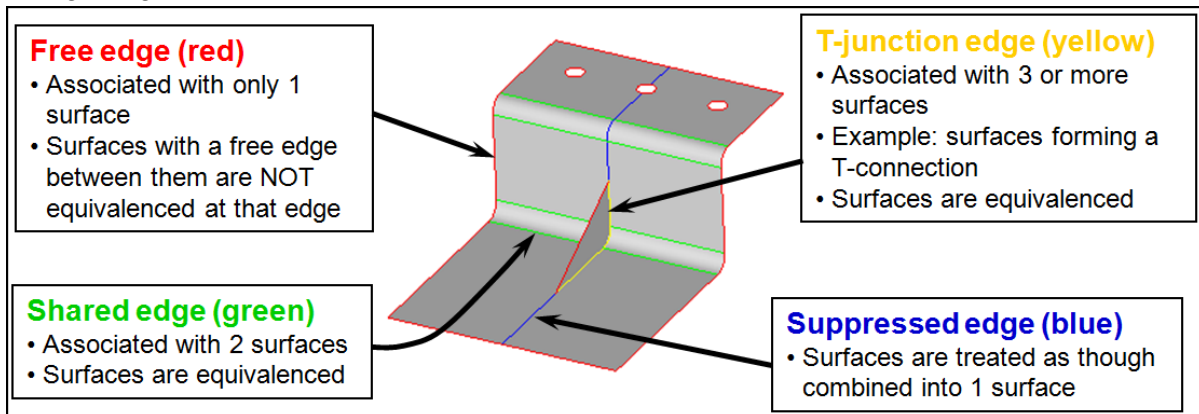
Automeshing

The Automesh function in HyperMesh allows for the rapid generation of a quality mesh on one or multiple surfaces. Within the automesh panel there are many options available which provide the user a high level of control over the ultimate mesh.



Topology Review

Automeshing of surfaces is dependent on surface topology, which is the connection of adjacent surfaces edges. Properly connected edges ensure a properly connected mesh. As discussed, in greater detail, in the previous chapter, topology is represented graphically through edge color.



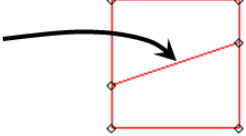
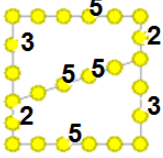
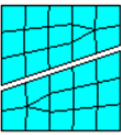
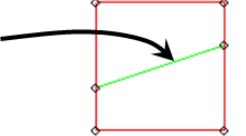
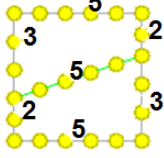
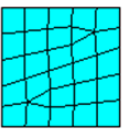
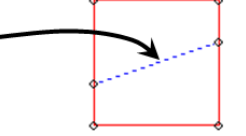
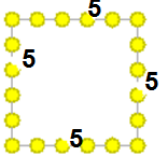
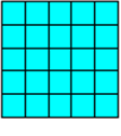
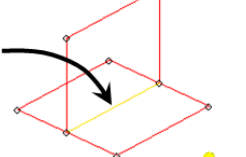
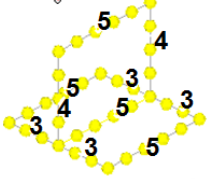
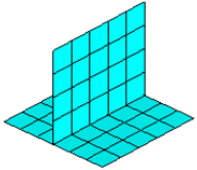
Mesh Connectivity

To properly represent a part with shell elements, those elements must be properly connected. Unconnected elements are in effect a slice in a part and all stresses, strains and deformations will stop at the unconnected region.

HyperMesh maintains edges in a part. This means that if you see a surface edge in the mesh (unless it is suppressed) you will see that same edge in the mesh. HyperMesh will place nodes along that edge and if the edge is properly connected then there will only be one set of nodes that will be shared by elements on either side of the edge. Node sharing between multiple elements is how elements are connected to each other.

Mesh Connectivity cont.

Below are examples of how edge topology affects the resultant mesh. There is an example for each of the 4 topological states (colors).

Free edges (red)	Shared edges (green)
<p>Free edge between 2 surfaces</p>  <p>Nodes are placed along edge for meshing</p>  <p>Mesh is discontinuous; nodes along the free edge are not equivalenced (Nodes only separated for illustration)</p> 	<p>Shared edge between 2 surfaces</p>  <p>Nodes are placed along edge for meshing</p>  <p>Mesh has proper connectivity; nodes along the edge are equivalenced</p> 
Suppressed edges (blue)	T-junction edges (yellow)
<p>Suppressed edge between 2 surfaces</p>  <p>Edge is ignored; nodes are not placed along the edge</p>  <p>Area is treated just like 1 surface; there is no line of nodes along the edge</p> 	<p>Non-manifold edge between 3 surfaces</p>  <p>Nodes are placed along edge for meshing</p>  <p>Mesh has proper connectivity; nodes along the edge are equivalenced</p> 

The Automesh Panel



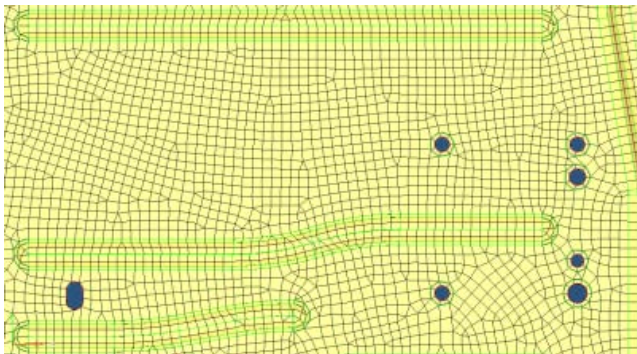
Within the automesh panel are numerous controls to allow for the creation of a quality mesh to individual specifications.

- Newly added to HyperMesh 10.0 are the **Align** and **Size** options on the Automesh panel.

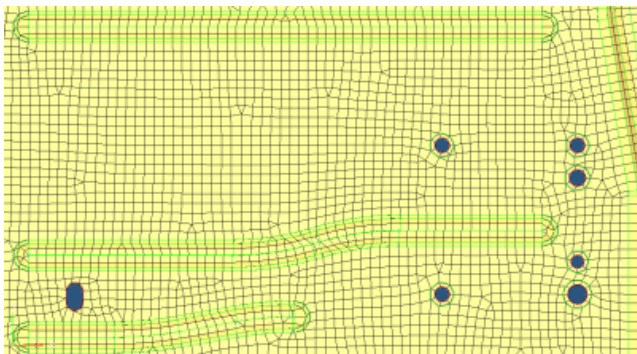
Flow:Align produces a more orthogonal quad dominated mesh

Flow:Size is active only when align is used and it enforces the global mesh element size with minimal min/max element size variation

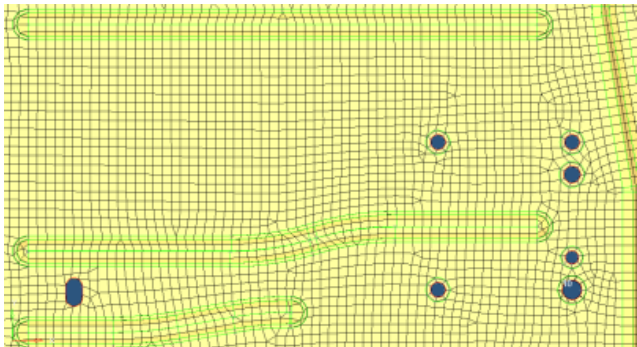
Mesh with no flow control



Mesh with Align Control

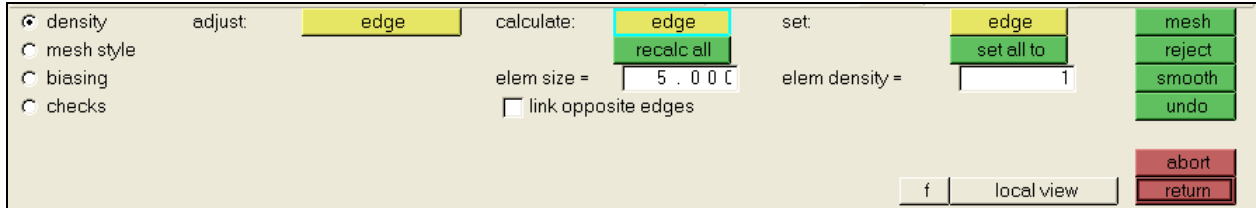


Mesh with Align and Size Control



Size and bias Subpanel

This subpanel is the most widely used and is the default for automeshing. Within this panel the user sets the desired element size and type, chooses options for flowing and mapping and then is provided with a preview mesh. From within the meshing module there are many options for mesh refinement.

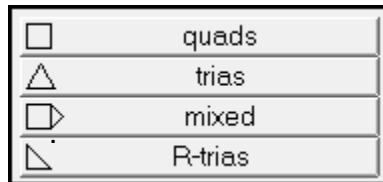


- **Density**

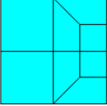
- **Adjust** – Left clicking on an edge will raise the element density on that edge by one, right clicking will lower the density by one. Clicking **mesh** will provide a new preview mesh with the changes.
- **Calculate** – Here the user can enter a new element size and either click on an edge to recalculate the density on that edge or click recalc all and change the element size for all the surfaces currently being meshed.
- **Set** – This option allows for the setting of element densities on a single or all edges to a user set number.

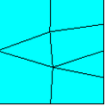
- **Mesh Style**

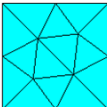
From this sub panel the user can indicated the type of mesh to create.

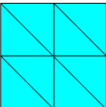


- ☐ **quads** – Quad dominant mesh; will use trias if necessary
- ☐ **trias** – Equilateral (60-60-60) tria mesh
- ☐ **mixed** – Quad mesh with use of trias to create as rectilinear a mesh as possible
- ☐ **R-trias** – Isosceles (45-45-90) tria mesh

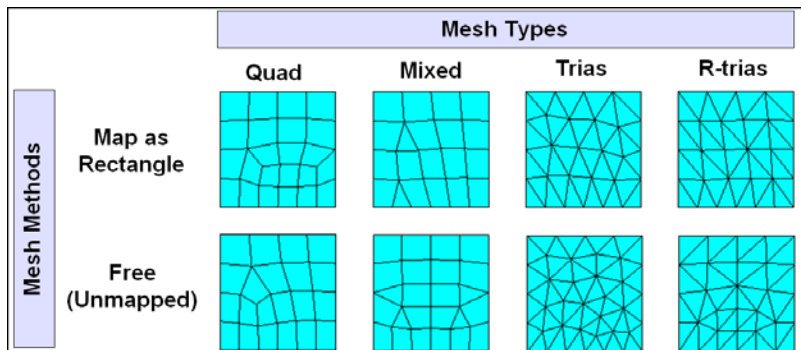
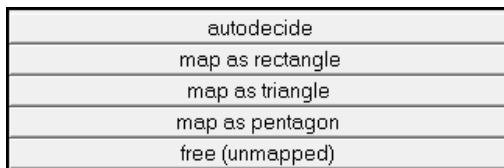

Quads


Mixed


Trias

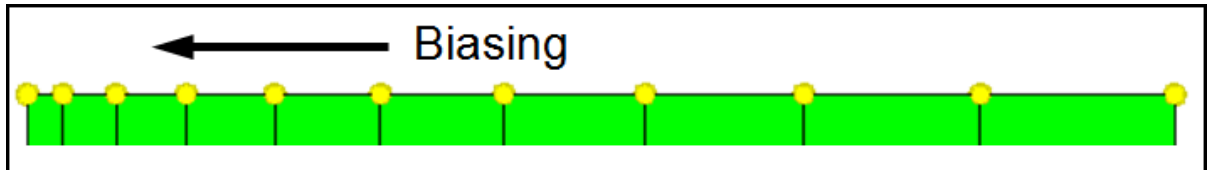

R-trias

This panel also allows the user to define the method or style of meshing. The “style” is the manner in which the mesh transitions between different density values.



- **Biasing**

The biasing subpanel allows the user to control the distribution of nodes during the nodes seeding by selecting biasing in the form of linear, exponential or bell curve distributions.



- **Checks**

The checks subpanel evaluates the quality of the generated mesh.

warpage	>	5 . 0 0 0	length	<	6 . 5 0 0
aspect	>	5 . 0 0 0	jacobian	<	0 . 7 0 0
skew	>	6 0 . 0 0 0			
quads:			trias:		
min angle	<	4 5 . 0 0 0	min angle	<	2 0 . 0 0 0
max angle	>	1 3 5 . 0 0 0	max angle	>	1 2 0 . 0 0 0
			f		local view

- **General Controls**

- **mesh** – Creates and updates the mesh preview using the current settings
 - User updates some mesh parameters on a surface, then clicks **mesh** to view the results
- **Accepting and Canceling**
 - **abort** – Cancels out of the meshing module; no mesh is created
 - **return** – Creates the mesh as per the mesh preview and returns to the automesh panel
- **f** **local view** – Offers Fit and controls from the View panel
 - Only applies to surfaces that are currently being meshed

☒ density
 ☐ mesh style
 ☐ biasing
 ☐ checks

adjust

edge

calculate:

edge

recalc all

elem size =
☐ link opposite edges

set

edge

set all to

mesh

reject

smooth

undo

f

local view

abort

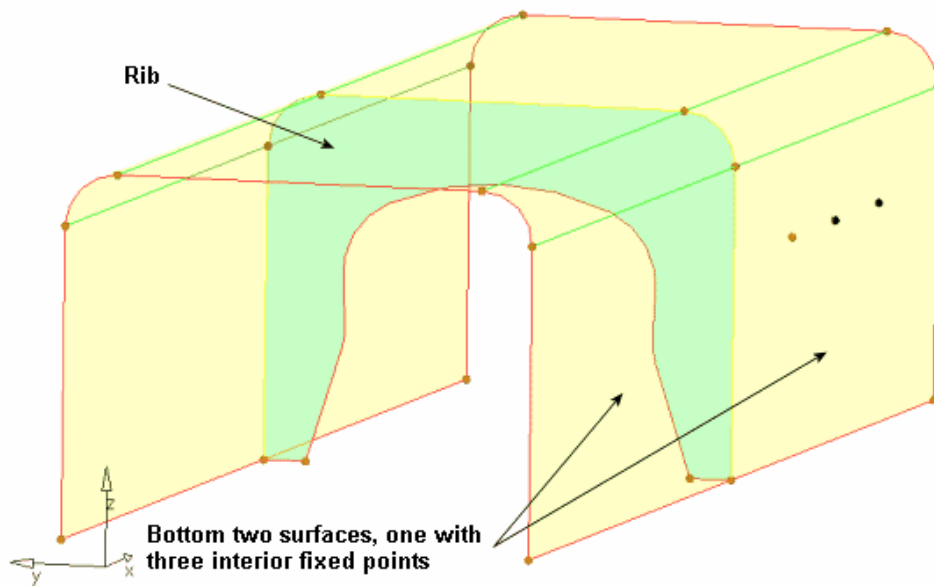
return



This toggle changes between automatic and interactive meshing modes. In automatic mode, HyperMesh will generate the mesh based upon the user entered values and options. In the interactive mode HyperMesh will generate the same mesh but will do so as a “preview” mesh allowing the user to use the above mentioned tools to manipulate the mesh before finalizing it.

Exercise: Meshing a Channel Bracket

This exercise uses the model file, `channel.hm`.



Step 1: Open and view the model file, `channel.hm`.

Take a few moments to observe the model using the different visual options available in HyperMesh (rotation, zooming, etc.).

Step 2: Mesh all the part's surfaces at once using an element size of 5 and the mixed element type (quads and trias).

1. Access the **automesh** panel in one of the following ways:
 - On the **Mesh** pull-down menu, select **Create**, and click **2D AutoMesh**
 - From the main menu, select the **2D** page, and then select **automesh**
2. Go to the **size and bias** sub-panel.
3. Select **surfs >> displayed**.
4. For **element size=**, specify 5.
5. Set the **mesh type:** to **mixed**.
6. On the panel's bottom-left corner, set **interactive** as the active mesh mode (it may currently be on automatic).
7. Check the message bar at the bottom and verify that the current component collector is Middle Surface.
8. Ensure that the elements to surf comp/elements to current comp toggle is set to **elems to current comp**.
9. Click **mesh** to enter the meshing module.


Notice that you are in the density sub-panel of the meshing module. There is node seeding and a number on each surface edge. The number is the number of elements that were created along the edge.

10. Click **return** to accept the mesh as the final mesh.

At this point, you could be done using the automesh panel to mesh the part. The mesh quality is very good. However, you will remain in the meshing module to perform the next steps, which demonstrate how to use various sub-panels to interactively control the creation of the mesh.

Step 3: Delete the mesh.

1. Access the **delete** panel in one of the following ways:

- On the **Collectors** toolbar, select **Delete** ()
- Press F2 on the keyboard

2. Switch the **entity** selector to **elems**.
3. Select **elems >> all**.
4. Click **delete entity**.
5. Click **return** to go back to the **automesh** panel.

Step 4: Mesh the surface having three fixed points interior to its surface.

You should still be in the **automesh** panel, **size and bias** sub-panel.

1. Leave all options in the menu panel as they are.
2. Select the surface that has three fixed points interior to its surface.
3. Click **mesh** to enter the meshing module.
4. Preview the mesh generated.

Step 5: Fit only the surface being meshed to the graphics area.

Fit the surface to the **Graphics** area in one of the following ways.

- On the **density** sub-panel, click **f**.
- Click **local view > fill** or click **local view** and pick **fill**.

Step 6: From the graphics area, specify a new element density along surface edges.

1. From the **density** sub-panel, click the selector, **adjust : edge** to make it active.
2. From the **Graphics** area, left-click an edge's element density number to increase it by one.
3. Right-click an edge's number to decrease it by one.

- Click and hold the mouse pointer on an edge's number and drag the mouse up or down to increase or decrease the number.
- Click **mesh** to update the preview mesh based on the change.

Step 7: From the menu panel, specify a new element density along surface edges.

You should still be in the **density** sub-panel.

- For **elem density=**, enter 10.
- Activate the selector **set : edge**.
- Click an edge's number to change its value to 10.
- Update the mesh by clicking **mesh** to preview the change.
- Change all edge densities to 10 by clicking **set all to**.
- Click **mesh** to preview the change.

Step 8: From the menu panel, specify a new element size to adjust element densities along surface edges.

You should still be in the **density** sub-panel.

- For **elem size=**, enter 7.
- Click **calculate: edge** to make it the active selector.
- Click an edge's number to calculate it based on an element size of 7.
The new number is calculated to create elements as close to a size of 7 as possible.
- Click **mesh** to preview the change.
- Click **recalc all** to base all edge densities on an element size of 7.
- Click **mesh** to preview the change.

Step 9: Change all edge element densities to reflect the initial element size of 5.

- For **elem size=**, specify 5.
- Click **recalc all**.
- Click **mesh** to preview the change.
- Return** to accept the mesh and go back to the **size and bias** sub-panel.

Step 10: Preview a mesh of the channel's rib.

You should still be in the **automesh** panel, **size and bias** sub-panel.

- With the **surfs** selector active, select the rib surface in the middle of the part.

2. Leave all options in the menu panel set they are.
3. Click **mesh** to enter the meshing module.
4. Preview the mesh generated.
5. Click local view and pick the rear view to display the rib's surface in this position, filled to the Graphics area.

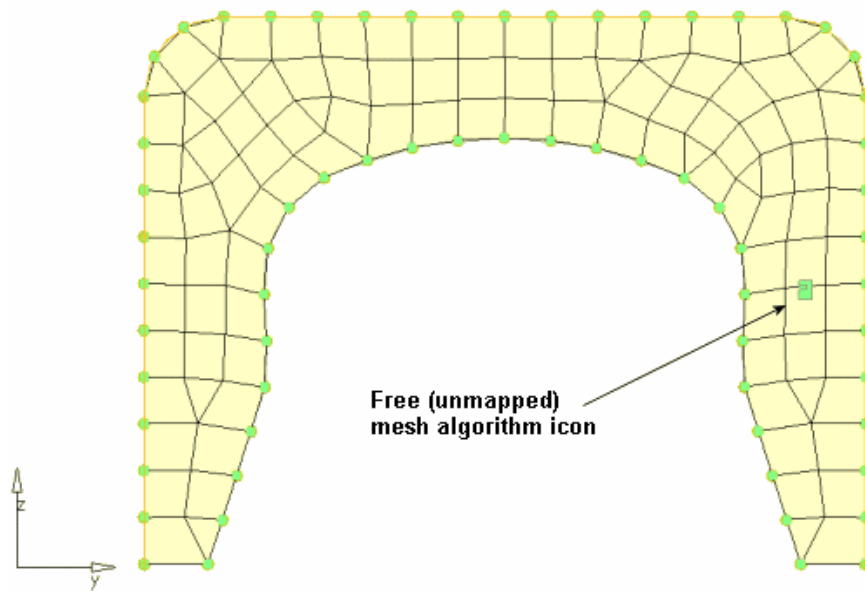
Step 11: Check the quality of the rib's preview mesh.

1. Go to the **checks** sub-panel.
2. Click **aspect** to identify all elements having an aspect ratio greater than 5.
3. Notice that no elements fail this check. In the status bar, the highest aspect ratio value reported is 1.72.
4. For **jacobian** specify 0.8.
5. Click **jacobian** to identify all elements having a jacobian less than 0.8.
6. Notice that several elements fail this check and are outlined red. The status bar reports the smallest jacobian value to be 0.71.
7. Change the **jacobian** check value back to 0.7.
8. Verify that no elements have a jacobian less than 0.7. (Click **jacobian**.)
9. Check for quad elements having a min angle less than 45.
10. Smallest angle is _____.
11. Check for quad elements having a max angle greater than 135.
12. Largest angle is _____.

Step 12: Change the rib's mesh pattern by changing the mesh method used for its surface.

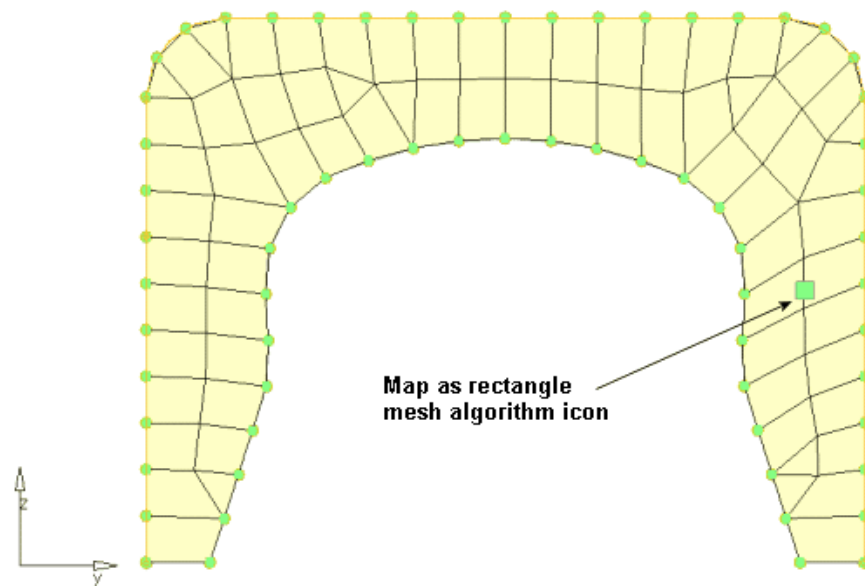
1. Go to the **mesh style** sub-panel.

- Notice that the edges' element density numbers disappear. Interior to the rib's surface is a white icon. It indicates the free (unmapped) mesh method is currently being used to mesh the surface.



Mesh on rib using the free (unmapped) mesh algorithm

- Under **mesh method**: set the option to **map as rectangle** (**autodecide** is default).
- Click **set all** found under **mesh method**.
- Notice that the icon changes to reflect the new mesh method.
- Click **mesh** to preview the change.



Mesh on rib using the map as rectangle mesh algorithm

Step 13: Check the quality of the rib's preview mesh again.

1. Go to the **checks** sub-panel.
2. Check for elements having an aspect ratio greater than 5.
Highest value reported is ____.
3. Check for elements having a jacobian less than 0.7.
Lowest value reported is ____.
4. Check for quad elements having a min angle less than 45.
Smallest value reported is ____.
5. Check for quad elements having a max angle greater than 135.
Highest value reported is ____.
6. Notice that in this case, the free (unmapped) mesh has better jacobian than the map as rectangle mesh.

Step 14: Change the rib's mesh method back to free (unmapped).

1. Go to the **mesh style** sub-panel.
2. Under **mesh method**: set the option to **free (unmapped)**.
3. Click **set all** found under **mesh method**.
4. Click **mesh** to preview the change.
5. Click **return** to accept the mesh as final and go back to the **automesh** panel.

Step 15: Preview a mesh of all displayed, unmeshed surfaces.

You should still be in the **automesh** panel, **size and bias** sub-panel.

1. Press V on the keyboard, and select **iso 1**.
2. Move your mouse off the pop-up menu to return to the **automesh** panel.
3. Accept all the default values.
4. On the menu panel's bottom-right side, click **failed surfs**.

The status bar displays the following message: "There are no surfaces with meshing errors". This is correct; all surfaces you selected to mesh so far have a mesh on them.

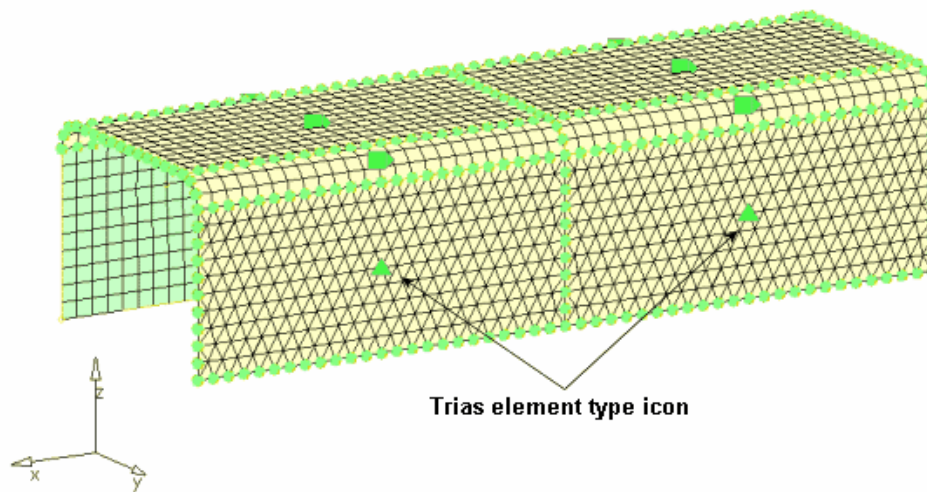
5. Click **unmeshed surfs** to identify and select all displayed unmeshed surfaces.
6. Click **mesh** to enter the meshing module.
7. Preview the mesh generated.

Step 16: Change the element type for some surfaces to trias.

1. Go to the **mesh style** sub-panel.
2. Click **toggle surf** found under **elem type**.

Notice that interior to each surface is a blue icon. It indicates the mixed element type (quads and trias) is currently being used to mesh the surface.

3. Under **element type**: select the menu **trias**.
4. Under **element type**: activate the selector **set surf.** (Click **set surf.**)
5. Left-click the blue icon twice in each of the channel's bottom two surfaces to set their element type to trias.
6. Click **mesh** to preview the change.



Preview of mesh with trias element type for the bottom two surfaces

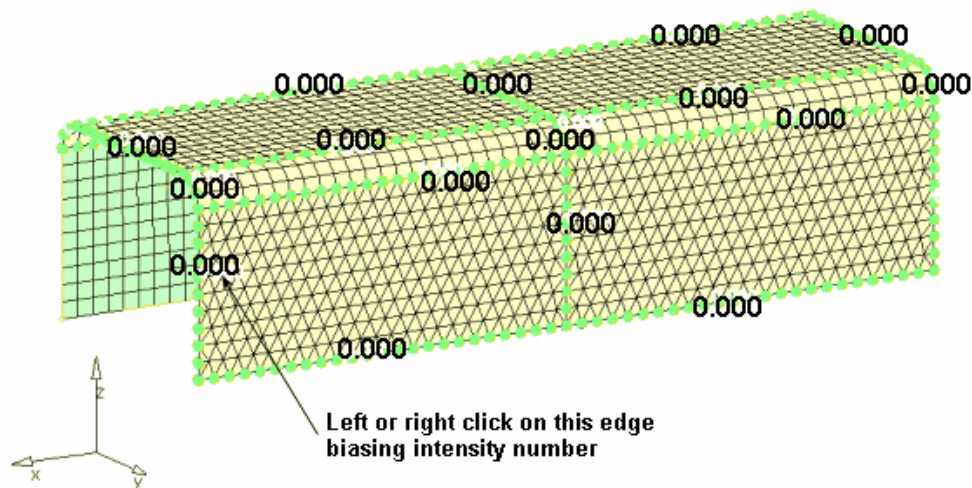
Step 17: Adjust the node spacing on surface edge (biasing).

1. Go to the **biasing** sub-panel.
2. Notice the bias intensity number (default 0.000) on each surface edge.
3. Leave the bias style set to **linear**.

This style corresponds to the positive slope of a straight line over the interval $[0,1]$ of the real line. For a positive bias intensity, smaller elements are at the start of the edge.

4. Verify that the selector **adjust: edge** is active.

- Left- or right-click the edge biasing intensity number as indicated in the image below to increase or decrease its value by 0.1.



Preview mesh in the biasing sub-panel

- Click and hold the mouse pointer on the same edge biasing intensity number and drag the mouse up to increase its value to 3.0.
- Click **mesh** to preview the change.
- For **intensity=** enter 10.
- Activate the selector, **calculate: edge**.
- Click the same edge's bias intensity value to change it to 10.
- Click **mesh** to preview the change.
- Switch **bias style:** to **bellcurve**.
This style distributes nodes along the edge in a pattern that is symmetric across the midpoint of the edge. For a positive biasing intensity, the smaller elements are at the start and end of the edge.
- Activate the selector **set: edge**, to make it active.
- Click the same edge's blue icon to change it from **linear** bias style to **bellcurve** bias style.
- Click **mesh** to preview the change.
- Click **return** to accept the final mesh and go back to the **automesh** panel.

Step 18: Remesh the channel's bottom two surfaces.

You should still be in the **automesh** panel, **size and bias** sub-panel.

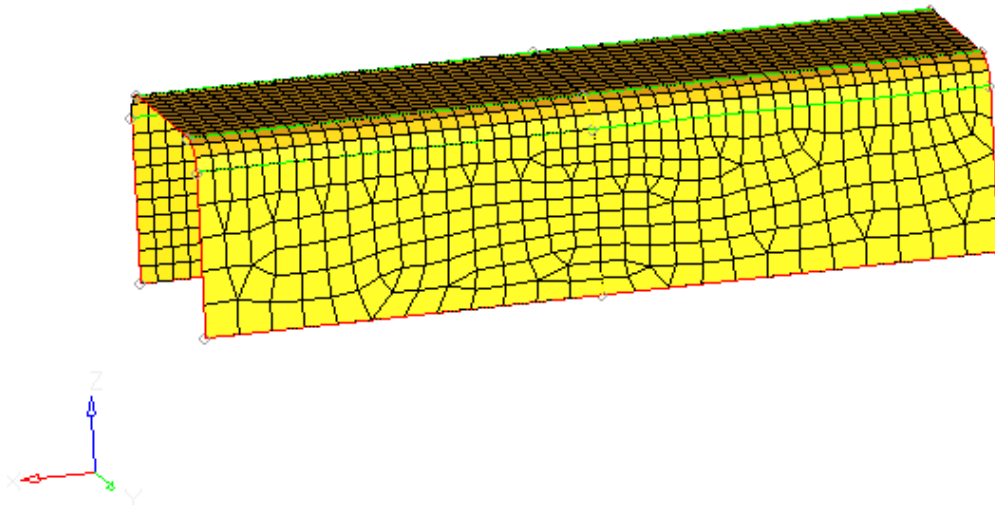
- On the panel's bottom-left side, switch the mesh mode from **interactive** to **automatic**.

This mode is not interactive; it does not take you to the meshing module. Rather, it meshes surfaces using only the basic parameters of the automesh panel. Interactive mode can be used to remesh the surfaces if you require the different options to control the created mesh.

2. With the **surfs** selector active, select the channel's bottom two surfaces (having the tria mesh).
3. Set **element size =** to 10.
4. Click **Mesh** to delete the existing mesh on the surfaces and create a new mesh.
5. Observe the resulting quad mesh on the remeshed surfaces.

Note that connectivity was maintained with the surrounding, smaller mesh. This is because the break connectivity option was not used.

6. Click **return** to go to the main menu.



Remeshed surfaces with element edge densities

Step 19 (Optional): Save your work.

Meshing of the channel part is complete. Now is a good time to save the model.

Section 2: Checking and Editing Mesh

Once a mesh is created, HyperMesh has many tools for checking element quality and modifying the mesh to make it more desirable. These tools can be used at almost any point in the meshing process. This section is separated from sections on creating mesh so that the focus can be on checking and editing tools.

In this section, you will learn how to:

- Identify shell element connectivity problems.
- Correct shell element connectivity problems.
- Review the model's shell elements to ensure connectivity problems were corrected.
- Re-mesh the elements along the rib.

Checking and Editing the Mesh: Tools

These mesh editing tools can be found on the 1D, 2D and 3D pages and can also be reached through the **Mesh>Edit>Elements** pull-down.

automesh	edit element	<input type="radio"/> Geom
shrink wrap	split	<input type="radio"/> 1D
smooth	replace	<input checked="" type="radio"/> 2D
qualityindex	detach	<input type="radio"/> 3D
elem cleanup	order change	<input type="radio"/> Analysis
	config edit	<input type="radio"/> Tool
	elem types	<input type="radio"/> Post

- **Edit element**
 - **Combine** – merge 2 or more elements into 1 single element
 - **Split** – draw a line to cut elements
 - **Cleanup** – interactively drag nodes to improve element quality
- **Split** – divide selected elements in a selected manner
- **Replace** – merge 2 nodes into 1 node (elements are now connected)
- **Detach** – disconnect selected elements from other elements
- **Smooth** – improves quality of selected elements
- **Quality index**
 - Set criteria for all standard quality checks
 - Simultaneously evaluate element quality
 - Combine all checks into a single value called “*composite quality index value*”
 - Edit nodes and elements interactively or by automatically maximizing element quality

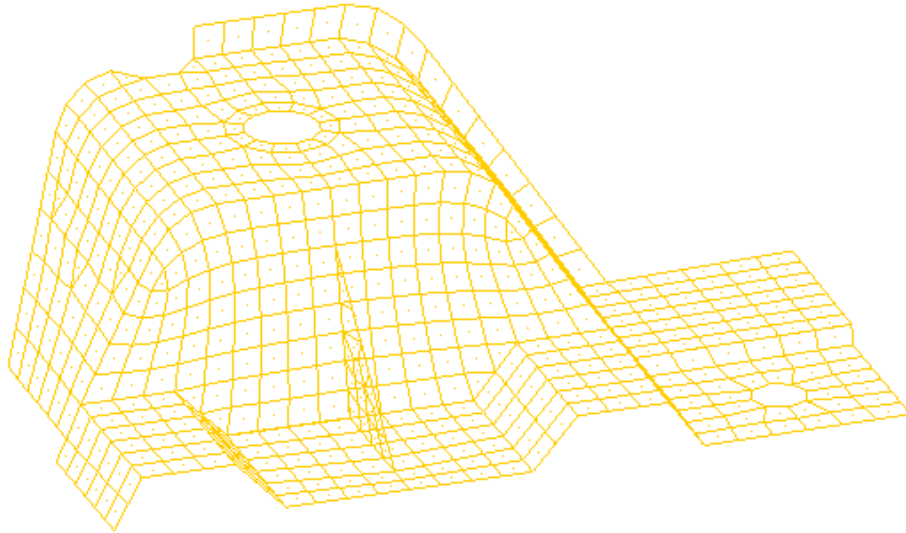
These tools can be found on the **Tool** page.

check elems	numbers	<input type="radio"/> Geom
edges	renumber	<input type="radio"/> 1D
faces	count	<input type="radio"/> 2D
features	mass calc	<input type="radio"/> 3D
normals	tags	<input type="radio"/> Analysis
dependency	HyperMorph	<input checked="" type="radio"/> Tool
penetration		<input type="radio"/> Post

- **Check Elms** – evaluate elements against various element quality criteria
 - Jacobian, warpage, angle, min. length, aspect ratio, skew
- **Edges**
 - Display free edges in the model by creating 1D elements on each one
 - Equivalence (merge) nodes within a specified tolerance
- **Normals** – display and correct element normals
- **Penetration** – display and correct element penetration

Exercise: Mesh Clean-up of a Plastic Cover

This exercise uses the model file, `cover.hm`.



Step 1: Retrieve and view the HyperMesh model file.

Open the file `cover.hm` located in the `<installation folder>\tutorials\hm` directory.

Step 2: Review the model's free edges to identify shell element connectivity problems.

1. Access the **edges** panel in the following ways:
 - From the **Pull-down** menu, select **Mesh**, then select **Check**, then **Components**, then **Edges**
 - From the main menu, select the **Tool** page, and then click **edges**
 - Press the SHIFT+F3 keys
2. With the **comps** selector active, click any element in the **Graphics** area.
The component containing the element is selected.
3. Click **find edges**.

Red, 1-D elements are displayed. They are organized into the new component named ^edges. A red 1-D element is created along each shell element edge that is free; one or more of the element edge's nodes is not shared by the adjacent elements.

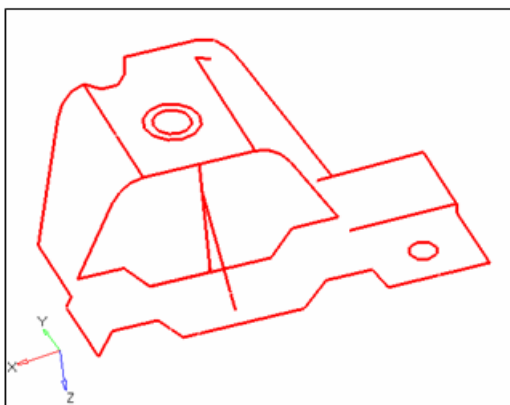
Note: For a component name whose first character is ^, the component and its contents is not written to the input file when the model is exported.

4. Click **Shaded Elements & Mesh Lines** ().

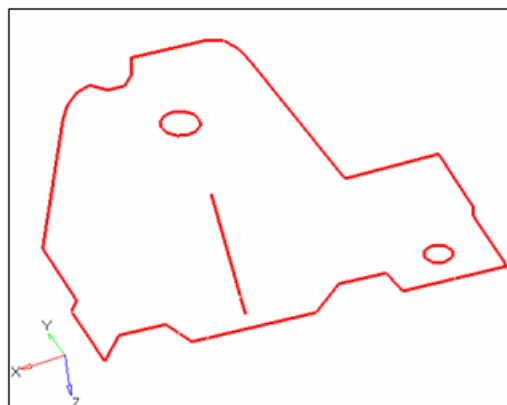
5. Observe the red, 1-D elements (free edges).
6. Try to identify gaps in the continuity of the mesh.

Hint: Look closely at free edges interior to the model.

7. In the model browser, turn the display on and off for the component, shells, to continue to identify which red, free edges do not belong.



All red, free edges




Red, free edges that belong

8. Turn on the display for the component, **shells**.

Step 3: Correct the shell element connectivity problems using the edges panel.

1. In the **tolerance=** field, type 0.01.
2. Click **preview equiv.**

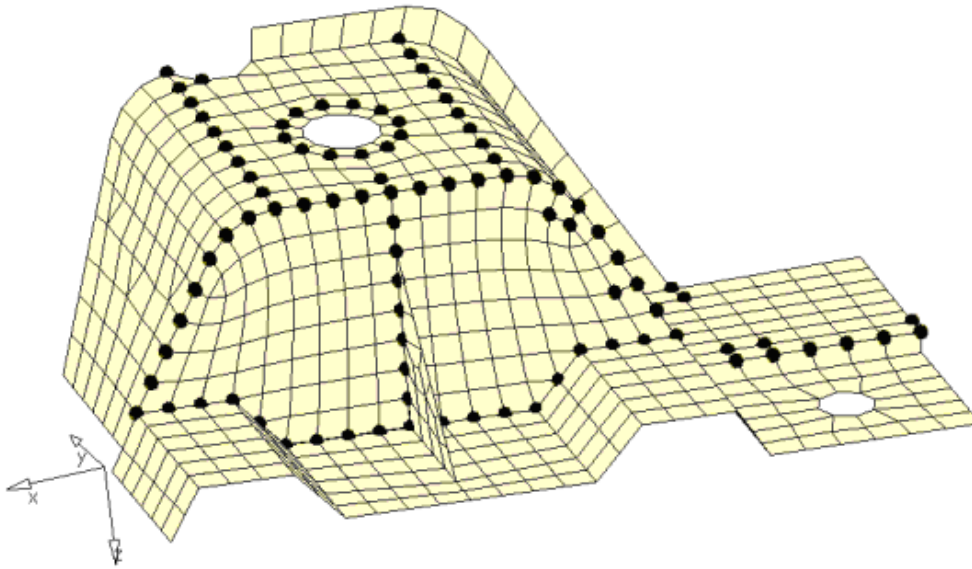
The **status bar** displays the following message, "81 nodes were found."

A sphere, , is created on nodes having a distance between each other equal to or less than the specified tolerance.

3. Notice that for this exercise's model, a sphere is not created on every node along all of the red, free edges, which do not belong. A larger tolerance must be specified to identify the rest of the nodes.
4. For **tolerance =**, increase its value until all 96 nodes are identified as shown in the following image.

Be careful not to increase the tolerance value to too much. Although the 96 nodes will be identified, an excessively large tolerance value may collapse elements when the identified nodes are equivalenced. To find out the maximum value that can be safely

used for tolerance without collapsing the elements, press the F10 key to go to the check elems panel, go to 2-d sub-panel and click on length. The status bar will display '... The min length is 1.49'. A tolerance value < 1.49 can safely be used, without causing any elements to collapse when identified nodes are equivalenced. Click return to go back to edges panel.



The nodes identified with preview equivalence

5. Click **equivalence**.

The 96 coincident nodes are equivalenced.

6. Rotate and observe the model to see that the mesh still looks as it should and no elements are collapsed.
7. Click **delete edges**.

The red, free edges and their component, **^edges**, are deleted.

8. Remain in the **edges** panel.

Step 4: Review the model's free edges again to confirm that all of the shell element connectivity problems have been corrected.

1. Click **find edges**.

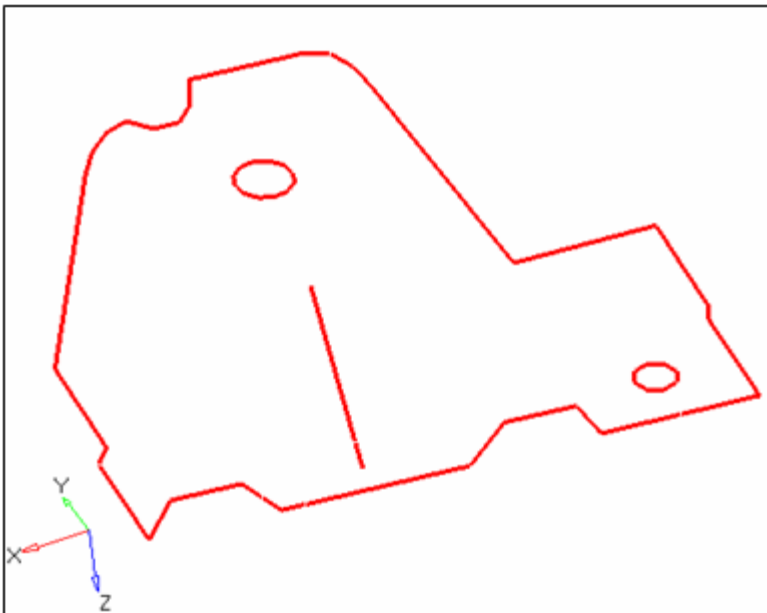
Observe the red, 1-D elements (free edges).

Are there any red, free edges that should not belong if the mesh were continuous or if all of the elements were connected?

Hint: Only red, free edges should exist on the perimeter of the part and on periphery of internal holes.

2. Use the model browser to turn the display off and on for the component, shells, to observe that all of the free, red edges belong.

3. After verifying that the model has correct red, free edges, click **delete edges**.



Red, free edges that belong

Step 5: Display the element normals and adjust them to point in the same direction.

1. Go to the **normals** panel. The **normals** panel can be accessed in the following ways:
 - From the pull-down menu, select **Mesh**, then **Check**, then **Elements**, and then **Normals**

- From the main menu, select the **Tool** page, then click **normals**

- Press the SHIFT+F10 keys

Choose the **elements** sub-panel and set toggle to **vector display normals**.

2. With the comps selector active, select one element from the graphics area to select the component.
3. Click **display normals**.

Arrows (vectors) are drawn from the element centroids and show the direction of the element normals.

Notice the arrows do not all point from the same side of the part. For some analyses, the element normals should point from the same side.

4. Click **size =** and enter the size which the normal should be in model units and select display normals again.

When **size =** is set to zero, the vector will be 10% of the screen.

5. Toggle **vector display normals** to **color display normals**.
6. Click **display normals**.

The element normals are displayed using colors. The red side of the elements is the positive normal direction, while the blue side is the negative normal direction.

7. Notice each side of the part shows red and blue.
8. Click the **orientation: elem** selector to make it active.
9. Select an element from the graphics area.
10. Click **adjust normals**.

All elements on either side of the part are the same color, red or blue.

The **status** bar displays the following message, "[X] elements have been adjusted."

If after adjusting the normals, there are still elements on one side of the part which are of different color, change to elems from comps for the entity selector, choose these elements and click reverse normals.

11. Click **return**.

Step 6: Review the quality of the elements using the check elems panel.

1. Access the **check elems** panel in one of the following ways:
 - From the **Pull-down** menu, select **Mesh**, then **Check**, then **Elements**, and then **Check Elements**
 - From the main menu, select the **Tool** page, then click **check elems**
 - Press the F10 key

2. Go to the **2-d** sub-panel.

3. Verify that **jacobian <** field is set to 0.7.

4. Click **jacobian** to determine if any elements have a jacobian of less than 0.7.

Elements having a jacobian of less than 0.7 are highlighted.

5. Notice that several elements on the triangular rib and around the smaller of the two holes have a jacobian of less than 0.7.

The **status** bar displays a message indicating how many elements failed this check.

6. In the graphics area, click an element.

A window displays that lists each quality check result for the element.

7. Click the right or left mouse button to close the pop-up window.
8. On the right side of the panel menu, switch from **standard** to **assign plot**.
9. Click **jacobian** to review again.

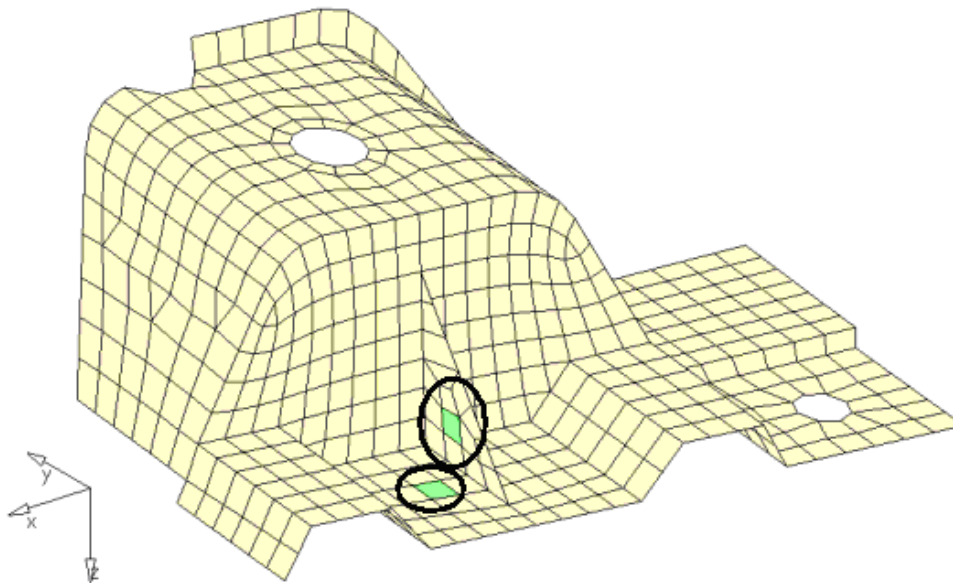
A legend for jacobian values appears and each element is colored accordingly. The red elements have a jacobian less than the threshold, 0.7.

10. Verify that **quads: min angle <** is set to 45.
11. Click **min angle** to determine if any quad elements have an angle of less than 45.
12. Notice that a couple of elements on the rib have an angle of less than 45.

13. Verify that the **max angle** >field is set to 135.
14. Click **max angle** to determine if any quad elements have an angle greater than 135.
15. Notice that several elements on the rib have an angle greater than 135.
16. Click **return**.

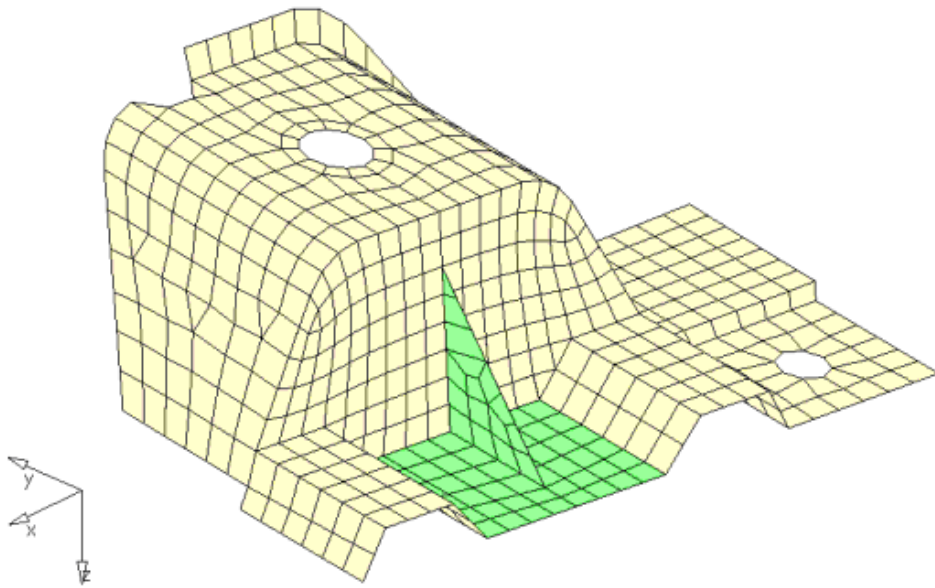
Step 7: Remesh the elements on the rib using the automesh panel.

1. Access the **automesh** panel in one of the following ways:
 - From the **Pull-down** menu, select **Mesh**, then **Create**, and then **2D Automesh**
 - From the main menu, select the **2D** page, then click **automesh**
 - Press the F12 key
2. Verify that you are in the **size and bias** sub-panel.
3. Switch the **entity** selector to **elems**.
4. Toggle to **interactive**.
5. For **element size=**, type 3.5.
6. Select one rib element from the graphics area.
7. Select one element on the plane of elements perpendicular to the rib and in the same plane as the rib's shortest edge as shown in the following image.



Example of elements to select

8. Select **elems >> by face** to complete the selection of elements as shown in the following image.

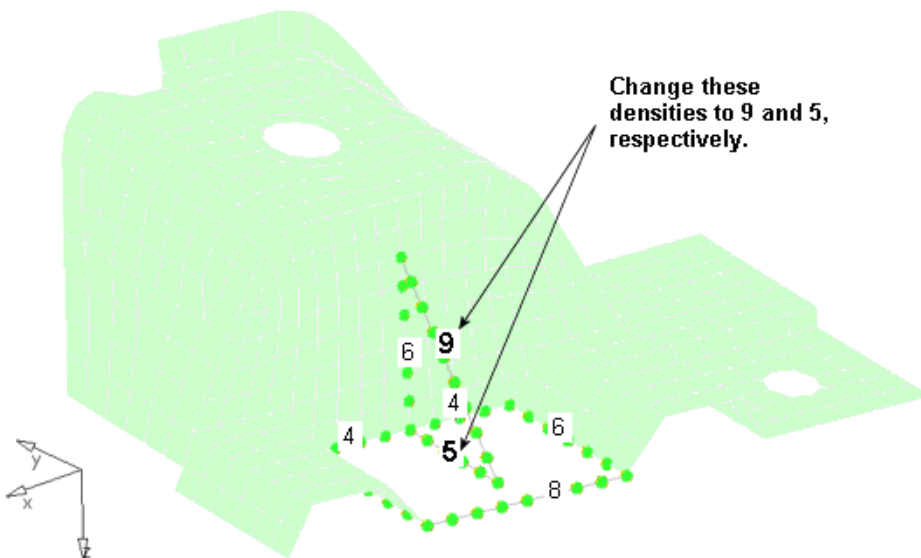


Elements selected using **by face**

9. Click **mesh**.

The meshing module appears.

10. In the **density** sub-panel, change the element density on the rib's hypotenuse edge to 9.
11. Change the element density on the rib's shortest edge to 5.
12. Keep all other element edge densities the same.



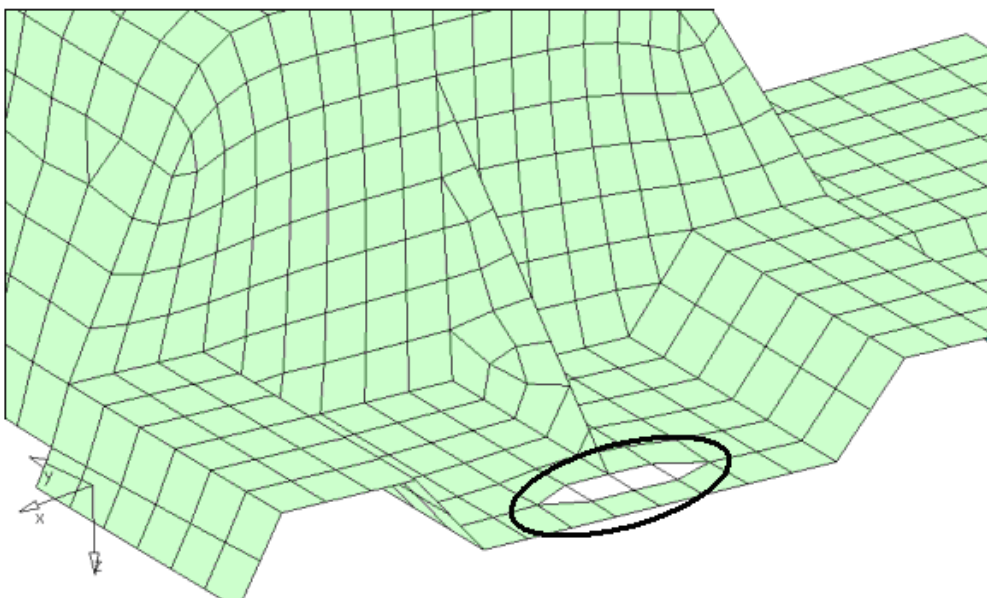
Adjusting element edge densities

13. Access the **mesh style** sub-panel.
14. Under **mesh method**, set the last option to **free (unmapped)**.

15. Under **mesh method**, select **set all**.
16. Click **mesh** to preview the mesh.
17. Go to the **checks** sub-panel, and check the jacobian, quads: min angle, and quads: max angle.
18. Notice that no elements fail the minimum and maximum angle checks.
Only a couple of elements have a jacobian of less than 0.7. The smallest jacobian is 0.68, which can still be considered good quality.
19. Click **return** to accept the mesh and go back to the main menu.

Step 8: Remove the tria elements from the rectangular plane of remeshed elements by first deleting the row of elements containing the trias.

1. On the **Tool** page go to the **delete** panel.
2. Ensure that **elems** is the active selector.
3. Select the elements as shown in the following image.
4. Click **delete entity**.
5. Click **return**.

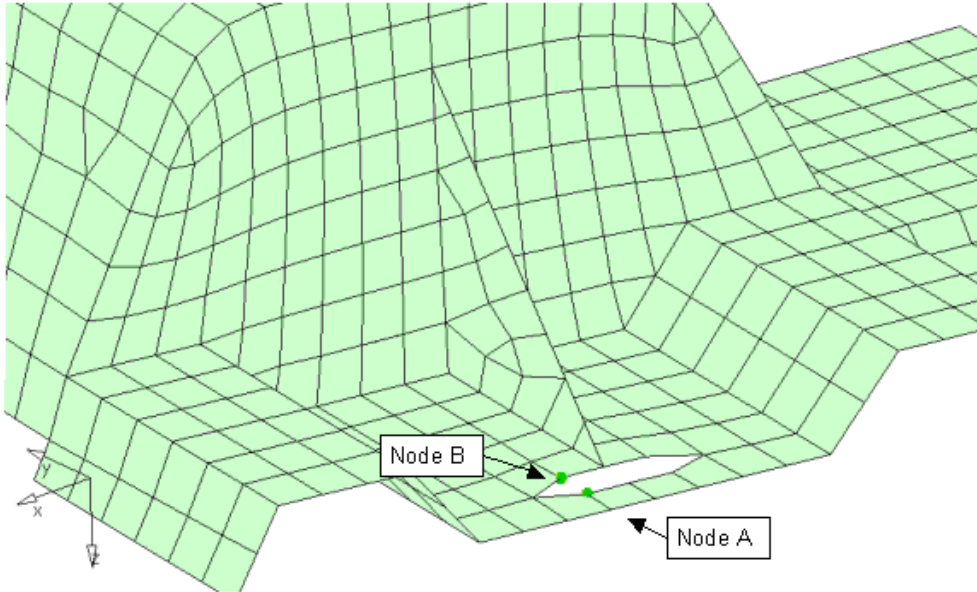


Elements to delete

Step 9: Stitch the gap that resulted from deleting the row of elements using the replace panel.

1. On the **2D** page, enter the **replace** panel and verify that just **equivalence** is checked.
2. With the **replace: node** selector active, select node A as indicated in the following image.
3. With the **with: node** selector active, select **node B** as indicated in the following image.

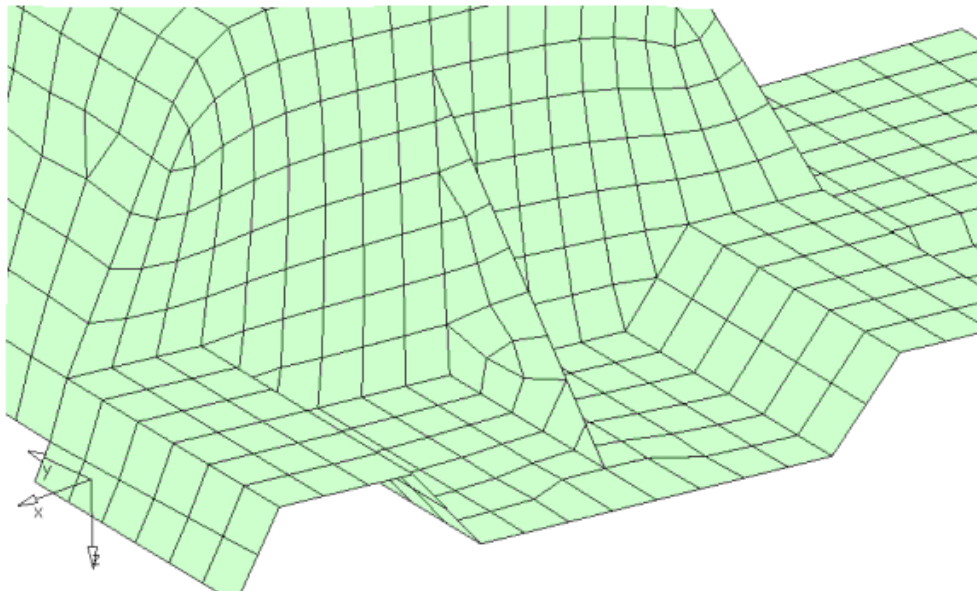
Node A is replaced with node B.



Select nodes A and B

4. Repeat #2 and #3 for the other nodes along the gap.

The resulting mesh should look like the mesh in the following image.



The mesh after replacing nodes on the bottom face

5. Click **return**.

Step 10: Use the smooth panel to adjust the node placement on the rectangular plane of remeshed elements.

1. On the **2D** page, enter the **smooth** panel.
2. Go to the **plates** sub-panel.
3. With the **smooth: elems** selector active, select an element on the rectangular plane of re-meshed elements.
4. Select **elems >> by face**.
5. For **iterations** = specify 10.
6. Switch the smoothing algorithm from **autodecide** to **shape corrected**.
7. Click **smooth**.
8. Click **return**.

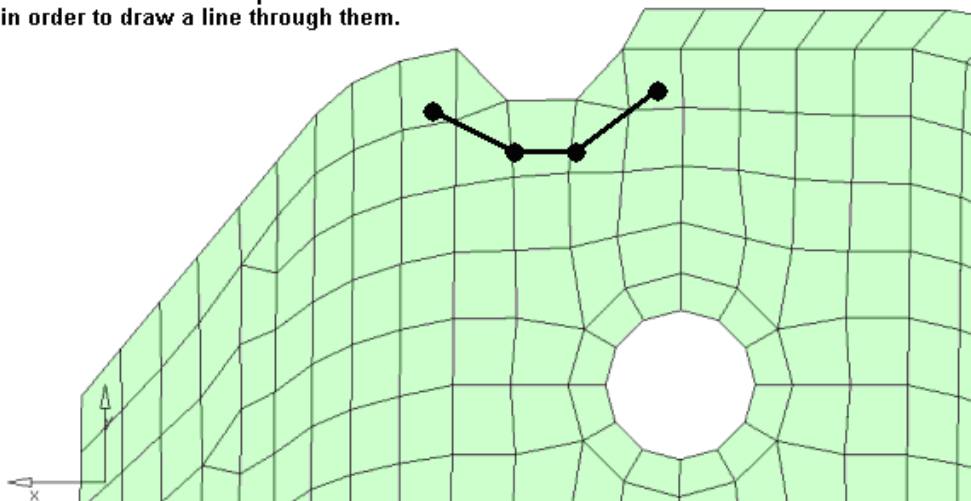
Step 11: Remove tria elements from another area of the model using the edit element panel, split and combine sub-panels.

1. On the **2D** page, enter the **edit element** panel.
2. Go to the **split** sub-panel.
3. With the **splitting line: points** selector active, click four screen points as shown the following image.

Temporary line segments are drawn to connect the points.

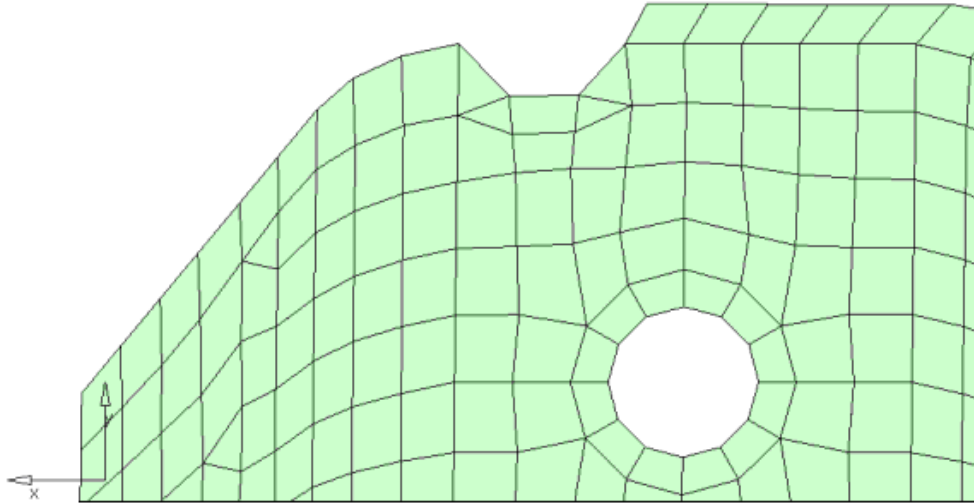
4. You can right-click to undo the last line segment drawn or you can click delete line to start over with selecting points.

Click at the four screen points indicated in order to draw a line through them.

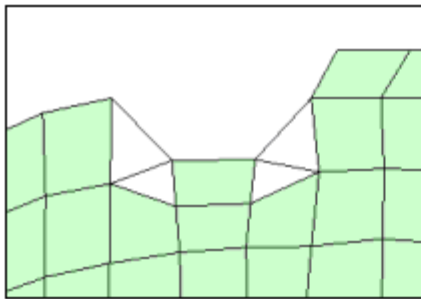


5. Click **split**.

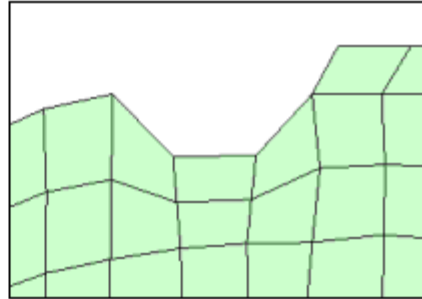
Elements that have the line pass through them are split. The resulting mesh should look like the mesh below. There are two pairs of adjacent tria elements.



6. Go to the **combine** sub-panel and set the toggle to **combine to quad**.
7. Select two adjacent tria elements as indicated in the following image.
8. Click **combine**.
9. Repeat # 7 and # 8 for the other two adjacent tria elements.
10. Remain in the **Edit Element** sub-panel.



Trias to select



Combining trias into quads

Step 12: Dynamically move nodes on the mesh area modified in the last step to improve element quality.

1. Go to the **cleanup** sub-panel and ensure it is set to **displayed elems**.
2. Click **cleanup**.

The element cleanup menu appears. It allows you to select various combinations of quality checks, specify a warning/unacceptable range for each check, and dynamically move nodes around to place them where you desire.

Elements are colored as follows:

- No color; appear blank: element passes quality checks
- Yellow: one or more quality check results falls into warning/unacceptable range
- Red: one or more quality check results are beyond unacceptable

Among the group of elements you modified in the last step, Step 11, two quad elements are red and one quad element is yellow.

3. Clear the **warpage** and **jacobian** check boxes, so that just the **max angle** is checked.
4. The model has 3 elements that are colored red.
5. With the node selector active, click an interior node of a red element and drag the node around until the element is no longer red.
6. Repeat # 4 for the other red elements.
7. Click **return**.

Step 13: For the same area of elements you focused on in the last step, optimize element quality by clicking nodes and elements.

1. On the **2D** page, go to the **qualityindex** panel.
2. With the **node optimize** selector active, click a few nodes of the mesh area you modified.

When a node is clicked, it is repositioned so that the elements attached to it have the best possible quality based on the criteria specified in the qualityindex panel.

3. Click the selector **element optimize** to make it active.
4. Click yellow and red elements in the same mesh area.

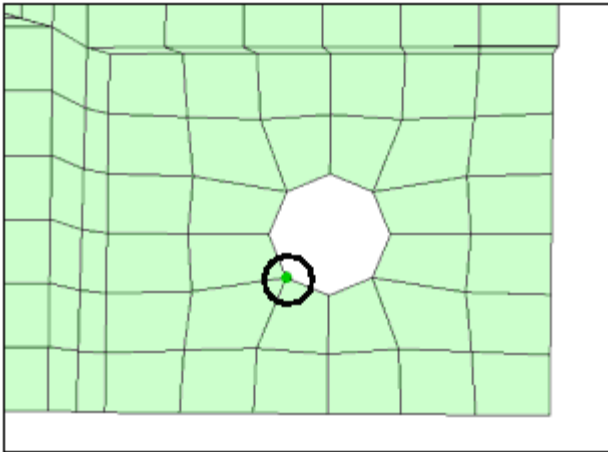
When an element is clicked, it is adjusted to have the best quality possible based on the criteria specified in the qualityindex panel.

When you click a red element, it may become yellow or black (no color assigned). When you click a yellow element, it may become black.

5. Click **return**.

Step 14: Add a ring of radial elements around the smaller of the two holes.

1. Click the **Utility** tab. If the Utility Menu is not displayed in the HyperMesh session, go to **View** menu and check **Utility** Menu
2. On the **Geom/Mesh** page, click **Add Washer**.
3. With the nodes selector active, select one node on the edge of the smaller hole as indicated in the following image.



Example node to select

4. Click **proceed**.

A pop-up window for the 'Add Washer ...' utility appears.

5. Toggle to **Width**, and for **Value** specify 3.0.
6. Select the **Minimum number of nodes around the hole** check box.
7. In the **Density**: field, enter 12.

Add Washer along a Circular Hole

Hole radius: 18.94

Number of layers: 1

Uniform

No	Selection	Value
1	Scale	1.0

☐ Create rigid spider along the hole

☐ Create local coordinate system

☐ Minimum number of nodes around the hole

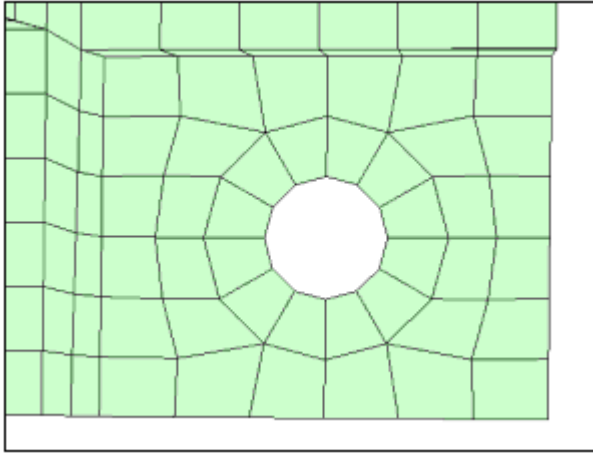
Density: 6

Add Reject Close

Add Washer dialog

8. Click **Add**.
9. Click **Close**.

The mesh around the hole should look like the mesh in the following image.



Resulting mesh around the smaller hole

Step 15 (Optional): Save your work.

With this exercise completed, you can save the model if desired.

Section 3: Batch Meshing

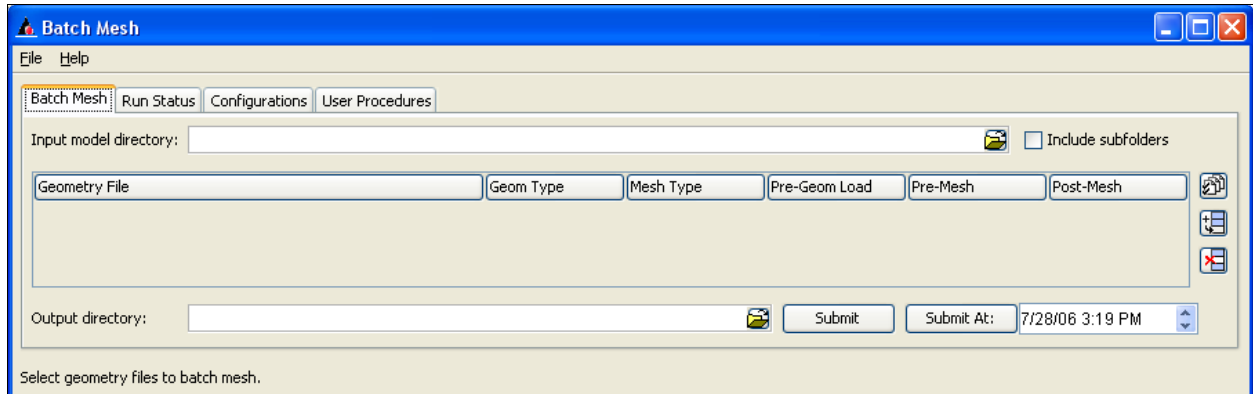
Geometry cleanup and meshing are often cited as time consuming aspects of finite element modeling. In HyperMesh, these tasks can be performed in batch mode with the Batch Mesher, requiring a minimum of input and user interaction. This section walks the user through the basic process of meshing a part using the batch mesher.

In this section, you will learn how to:

- Define a configuration for the batch mesh
- Edit the criteria and parameter files
- Run the batch mesh job
- Monitor and review the batch mesh job

Batchmesher

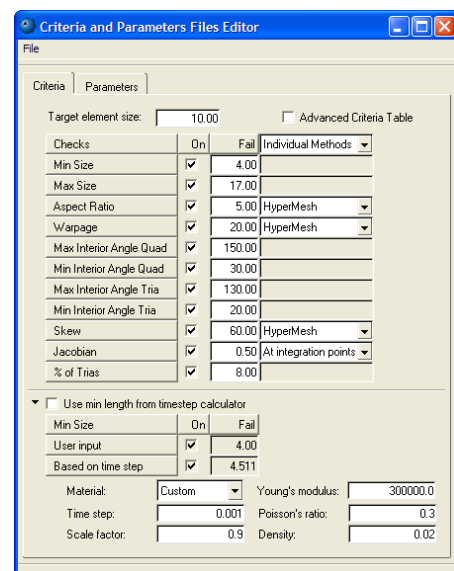
- Performs geometry cleanup and automeshing in a batch mode
 - Can dramatically improve productivity since there is minimal user interaction (especially for large assemblies)
- Uses *criteria* and *parameter* files to determine how the parts should be meshed
- Can mesh multiple files in the same run



Batch Meshing Procedure

1. Create / specify *Configurations*

- Combination of element criteria and geometric parameters.
 - Element criteria are element quality requirements.
 - Geometric parameters are other requirements (element type, removal of pinholes, removal of fillets, etc.)
- Criteria and parameters can be edited with the *Criteria and Parameters Files Editor*.



2. Register and specify *User Procedures* (optional)

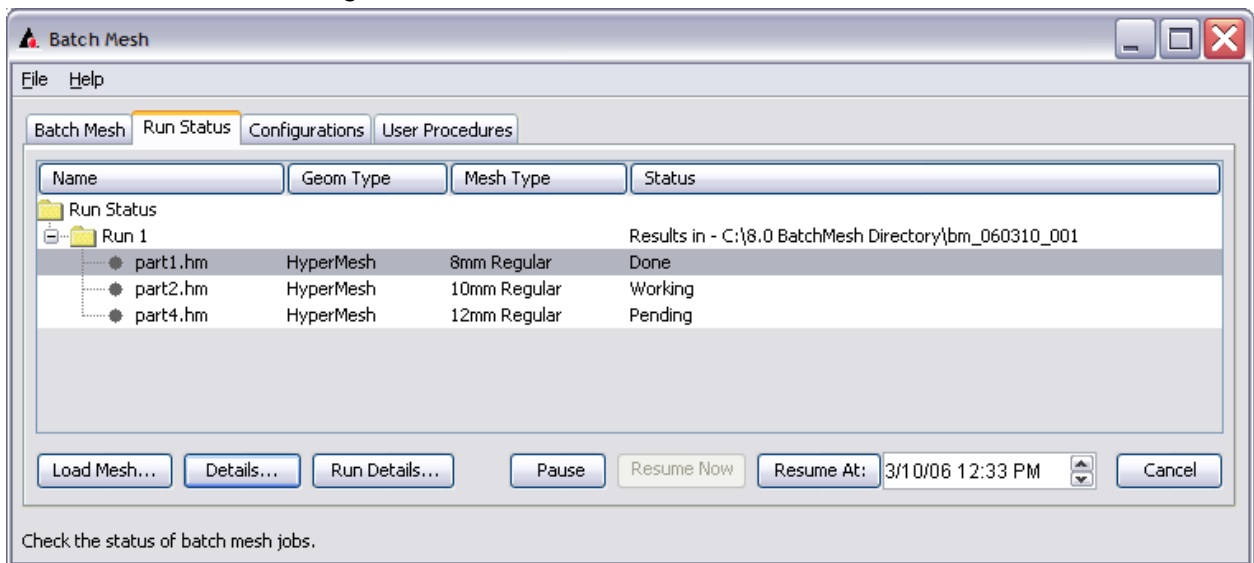
- Custom TCL scripts (macros) created by the user
- Performs additional operations on models during the batch run
- *Pre Run* and *Post Run* options can specify procedures performed before or after the run
- Can specify when the procedure should be performed during the run
 - *Pre-geometry load* – as soon as batch mesher is invoked
 - *Pre-batch mesh* – just before loaded geometry is meshed
 - *Post-batch mesh* – after meshing is finished

3. Set up the *Batch Mesh* job

- Specify a directory where geometry files are located
- Select geometry files from the directory
- Select a configuration to use for the mesh type for each file
- Specify any user procedures to be performed on each part
- Specify a directory where the meshed files will be located
- Run the batch mesh job




4. Monitor the *Run Status* of the job

- Use the *Run Status* tab to keep track of all your submitted jobs
- Use *Load Mesh* to open a selected file in a new HyperMesh session to view the resulting mesh



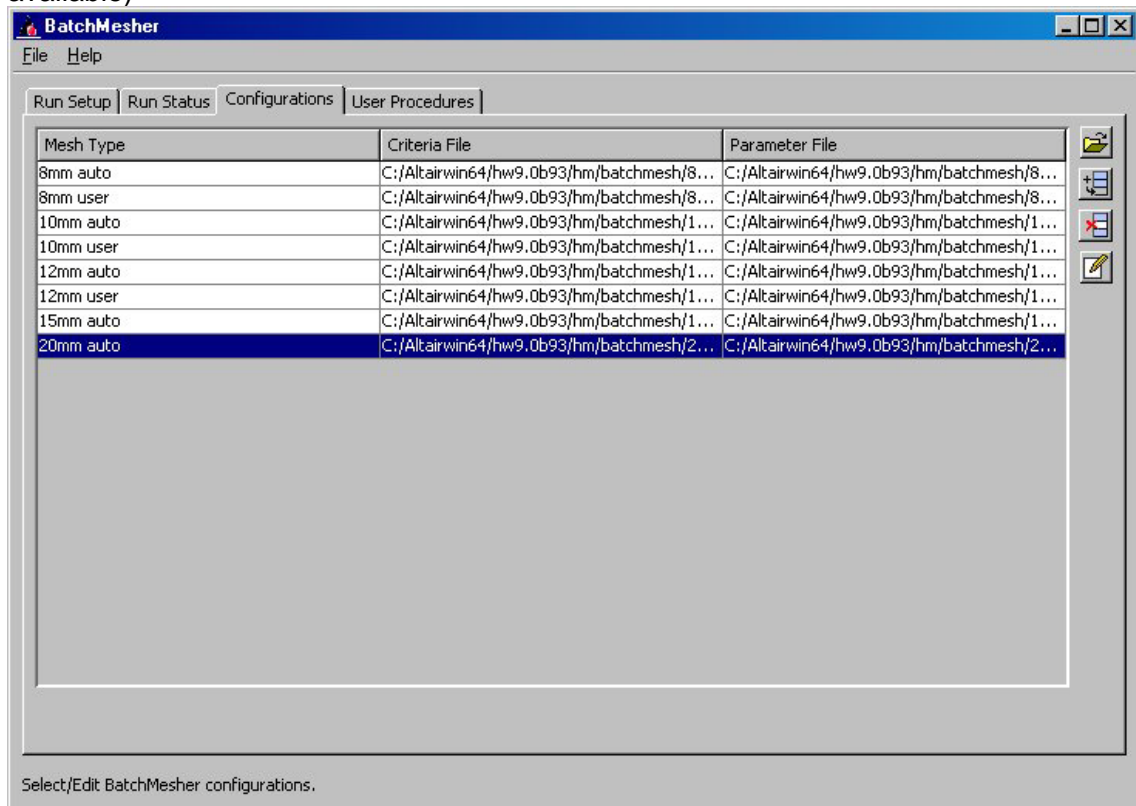
Exercise: Batch Meshing a Shell Bracket




Step 1: Open Batch Mesher.

1. Click **Start**, point to **Programs**, point to **Altair HyperWorks 9.0**, and click **Batch Mesher**.
2. In the **Input Model Directory** field, click **Open** () to browse to the appropriate directory.
3. In the **Output Directory**, click **Open** () to browse to the appropriate directory if different from the input model directory. If desired, you may request the .hm files be returned to the input model directory.
4. Click **select files**, .
5. For **Type of Geometry**, select the appropriate CAD data type. For this exercise, we will use geometry already loaded into Hypermesh.
A filter will help select applicable models for batch meshing to HyperMesh.
6. Locate and select the following model files, (use the CTRL key to select multiple files):
part1.hm, part2.hm and bm_housing.hm.
7. Click **select**.

Step 2: Define a configuration for the batch mesh run.



1. Click the **Configurations** tab. (Note there are several common configurations already available)



2. Click **Add Entry**, .
3. In the **Mesh Type** field, type a name for the mesh type.
4. Select the **Criteria File** field, and click **Find Criteria Param File**, .
5. Select the file `bm_housing.criteria`.
6. Select the **Parameter File** field, and click **Find Criteria Param File**, .
7. Select the file `bm_housing.param`.

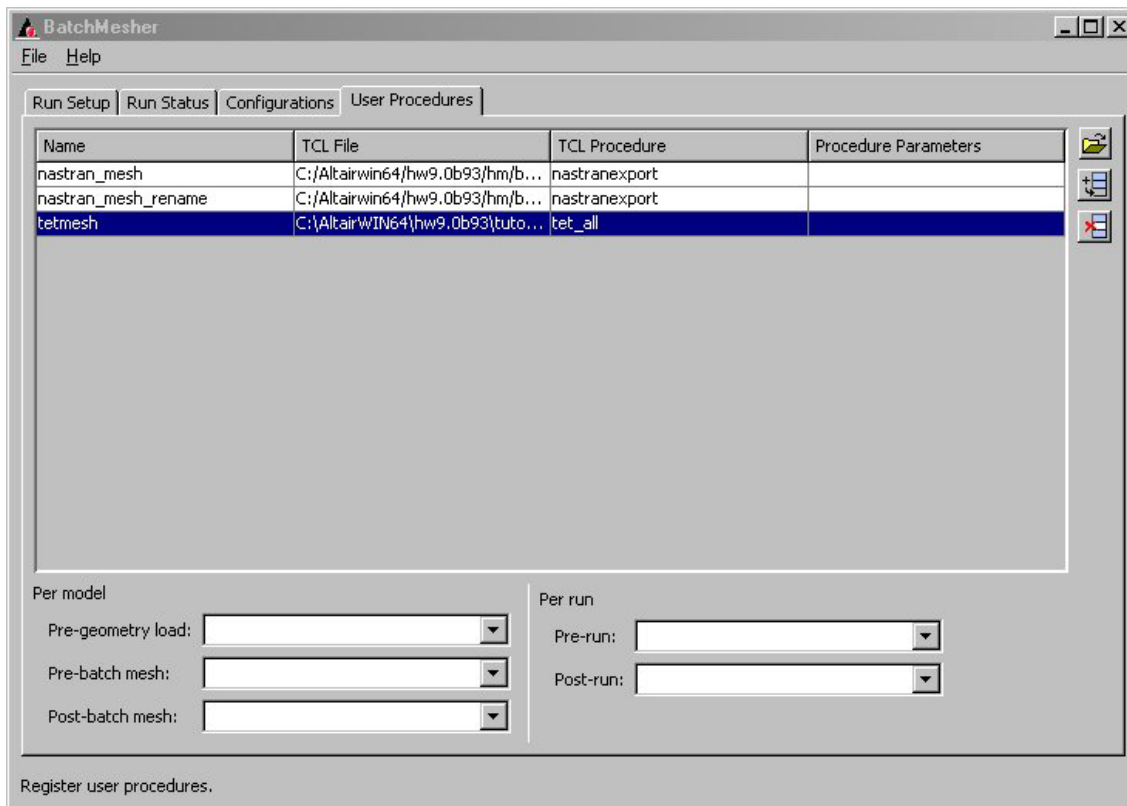
A new mesh type is now available for selection on the **Run Setup** tab.

Step 3: On the User Procedures tab, set up a simple script to perform a tetramesh on the housing.

1. Click **Add Entry**, .
2. Select the **TCL File** field, and click **Find TCL File**, .
3. Select the file `hm_housing.tcl`.
4. In the **TCL Procedure** field, select the procedure `tet_all`.

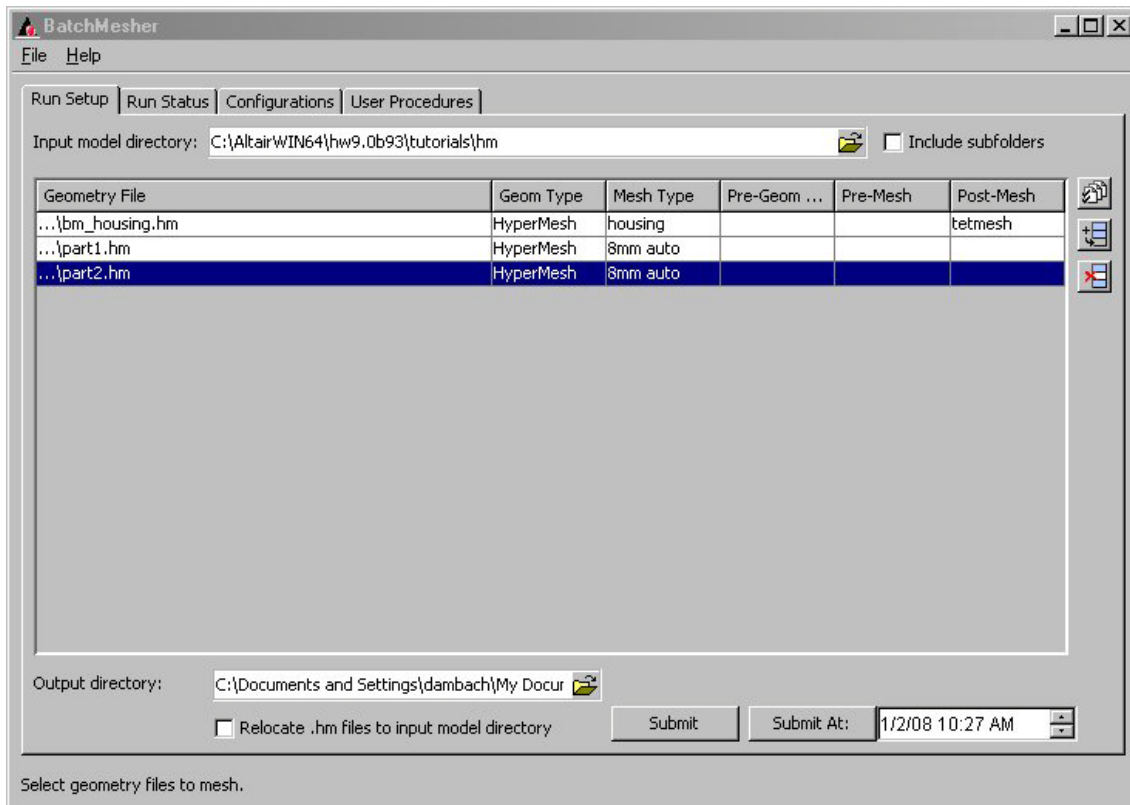
5. In the **Name** field enter **tetmesh**.

A new post-batch meshing script is now available for selection on the **Run Setup** tab.



Step 4: On the **Run Setup** tab, begin defining a configuration for the batch mesh run.

1. In the **Mesh Type** field, select a predefined mesh type for each of the parts as follows:
 - bm_housing.hm: use the mesh type you defined in Step 2.
 - part1.hm: use 8mm auto.
 - part2.hm: use 8mm auto.
2. From the **Post-Mesh** drop-down list for the housing, select the procedure **tetmesh**.
This will run the tetramesher on all available shell elements after batch meshing.



- Click **Submit** to initiate the run.

or

Click **Submit At** to submit the job at a specified time.

The application automatically switches to the **Run Status** tab. As the parts run, the status changes from **Pending** to **Working** to **Done**.

- Once the part is at the **Working** state, select the part and select **Details**.

A detailed summary appears with the status of the model through its Batch Mesher steps, the overall failed elements, and quality index.

- Once the part is at the **Done** state, click **Load Mesh** to load the mesh into HyperMesh for model interrogation.

- Once all parts have been meshed, select **Run Details** to obtain an overall run status.

Any file can be paused or canceled. If the file is paused, it can be resumed immediately or at a specific time.

Once the Batch Mesher session has been setup with file directories and mesh types, it can be saved as a config that can be loaded at a future time.

It is also possible to load an entire set of models that has already been batch meshed in order to take advantage of the **Load Mesh** option in the **Run Status** tab.

If you choose to make modifications to your criteria or parameter files, the run can be submitted again. The new files will be placed in a separate sub-directory.

Step 5 (Optional): Edit the Criteria and Parameter Files.

1. Go to the **Configurations** tab.
2. Click **Mesh Type**.
3. Click **Edit File**.

The Criteria and Parameter Editor window appears.

Criteria and Parameters Files Editor - [C:\Altair\WIN64\hw9.0b93\tutorials\h...

File

Criteria Parameters

Target element size: 6.000 ☐ Advanced Criteria Table

Checks	On	Fail	Individual Methods
Min Size	<input checked="" type="checkbox"/>	2.000	
Max Size	<input type="checkbox"/>	13.000	
Aspect Ratio	<input checked="" type="checkbox"/>	4.000	HyperMesh
Warpage	<input checked="" type="checkbox"/>	15.000	HyperMesh
Max Interior Angle Quad	<input checked="" type="checkbox"/>	130.000	
Min Interior Angle Quad	<input checked="" type="checkbox"/>	50.000	
Max Interior Angle Tria	<input checked="" type="checkbox"/>	110.000	
Min Interior Angle Tria	<input checked="" type="checkbox"/>	25.000	
Skew	<input checked="" type="checkbox"/>	40.000	HyperMesh
Jacobian	<input checked="" type="checkbox"/>	0.750	At integration points
% of Trias	<input checked="" type="checkbox"/>	15.000	

☐ Use min length from timestep calculator

	On	Fail
Min Size		
User input	<input checked="" type="checkbox"/>	2.000
Based on time step	<input checked="" type="checkbox"/>	4.511

Material: Custom Young's modulus: 300000.0

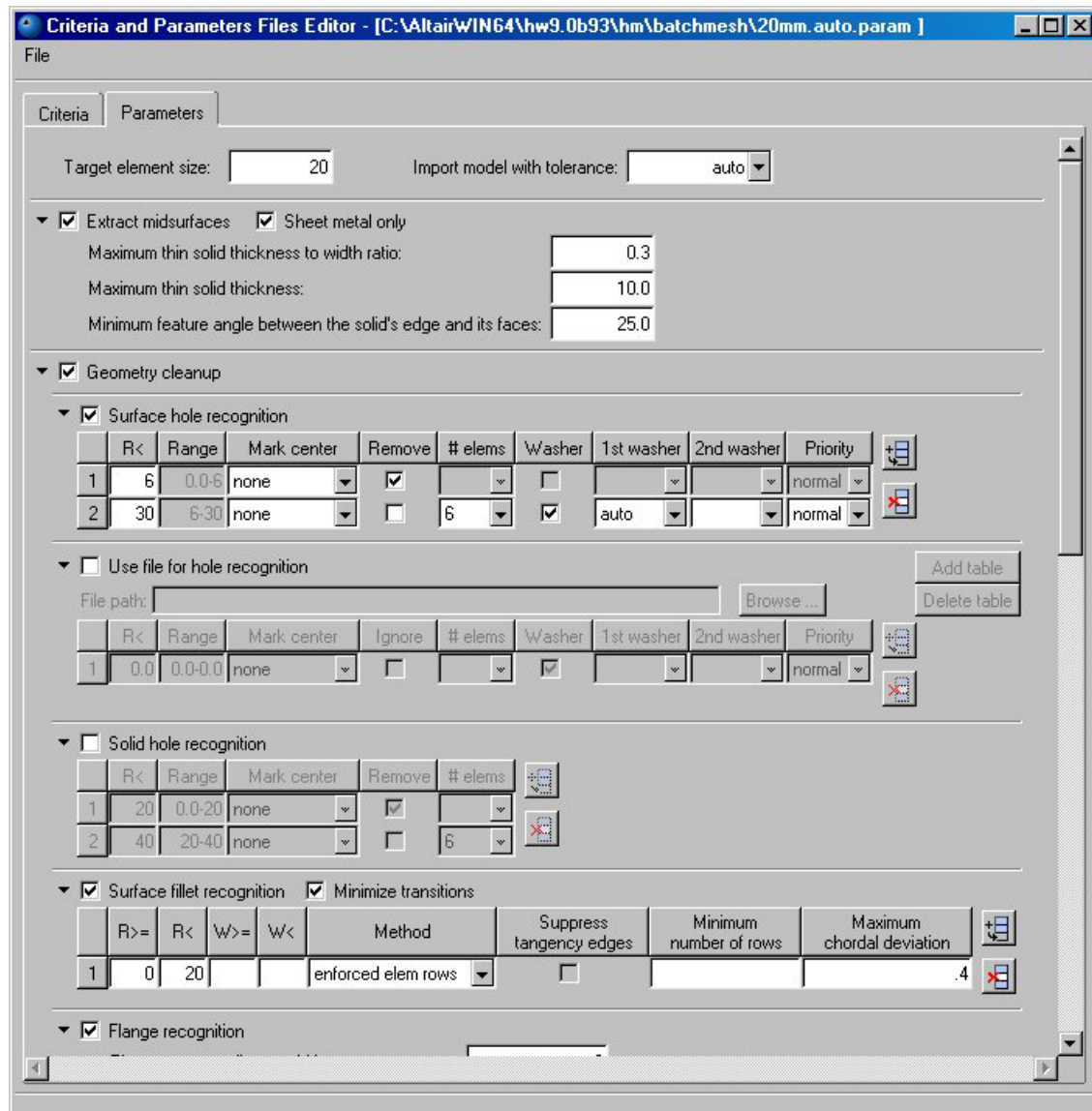
Time step: 0.001 Poisson's ratio: 0.3

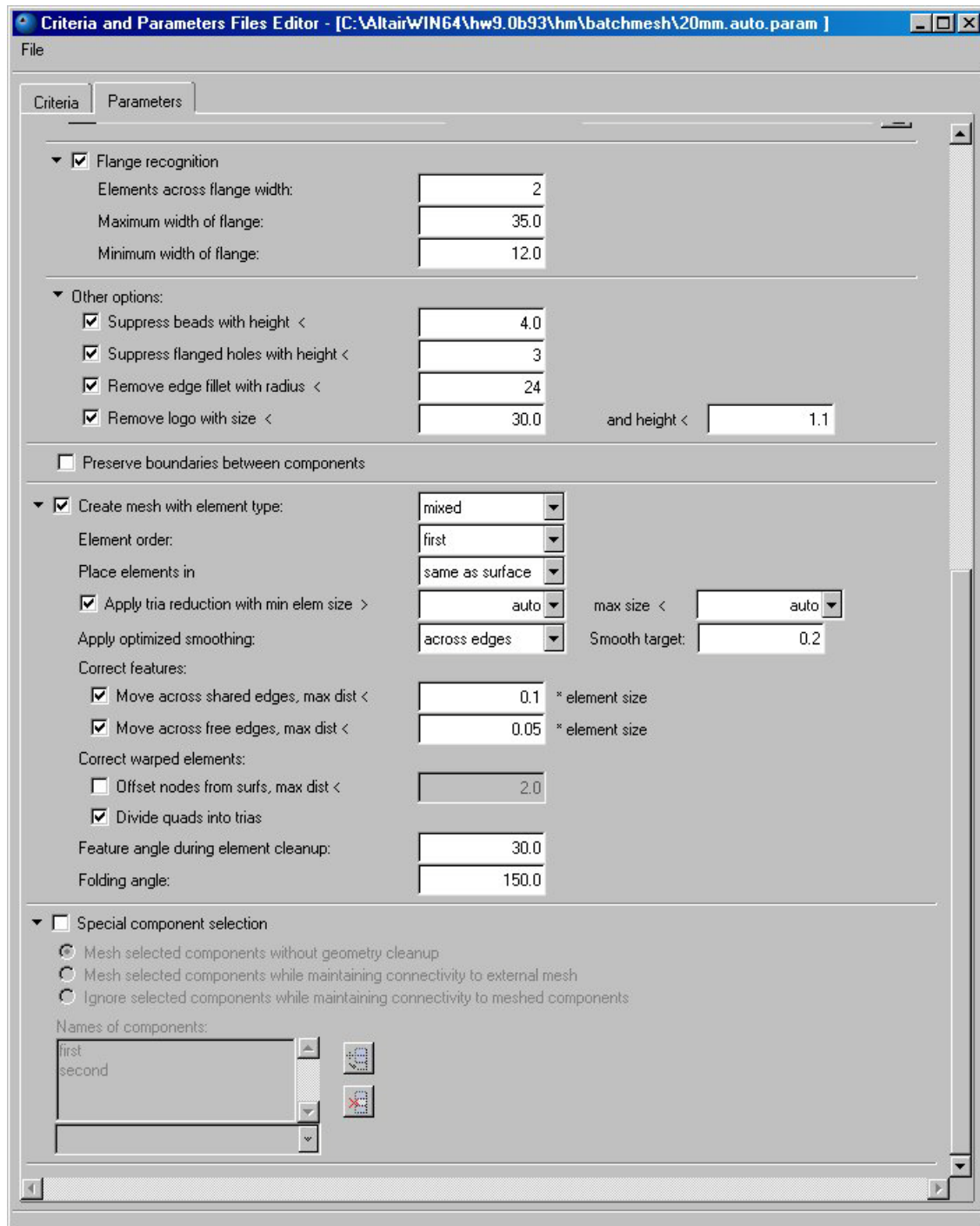
Scale factor: 0.9 Density: 0.02

On the **Criteria** tab, you can set the target element size, element criteria, and the method that is used to calculate the values. An advanced criteria option can be enabled to give more control over the intermediate QI values, but is usually not necessary in order to obtain a good quality mesh.

A timestep calculator is also available for explicit solver models. If this option is selected, the overall minimum element size will be set by this option and will gray out at the top of the element checks.

The **Parameters** tab sets all of the meshing control over various geometric features.





Chapter 4

Solids and Hexas

Section 1: Creating and Editing Solid Geometry

HyperMesh has several functions that require the definition of a volume, such as creating tetrahedral and hexahedral meshes. This can be done either with surfaces that enclose the volume, or with solid geometry entities. Working with solids provides a couple of advantages over surfaces. Selecting the volume for the function requires only a single click because solids represent the volume with a single entity, as opposed to surfaces. Solids that are topologically connected to each other also allow the functions being used to recognize the connection. Creating mesh in these cases allows the mesh in adjacent volumes to automatically have proper connectivity.

In this section, you will learn:

- What is solid geometry
- What is topology
- What does 3D topology look like

Solids are geometric entities that define a three-dimensional volume. Geometric entities are defined as follows:

- Point: 0 dimensional; has only x, y, and z coordinates
- Line: 1-dimensional; has length, can be curved through 3-dimensional space
- Surface: 2-dimensional; has an area
- Solid: 3-dimensional, has a volume

Solid Geometry

HyperMesh supports the same solid geometry that is created in popular CAD software. Solid geometry can be achieved through the importation of native CAD data or can be easily created from bounding surfaces using the **Bounding Surfaces** option from the **Solids** panel on the **Geom** page.

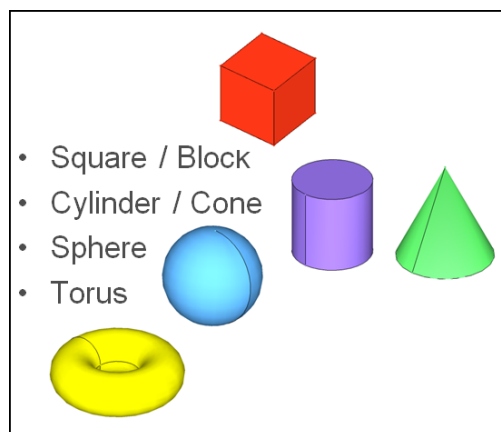
Additionally HyperMesh has numerous solid geometry creating tools to assist in the manipulation of solids. These can be found through the pull-down Menu

Geometry>Create>Solid and include many options for creation of solids through

- Dragging – Extrusion of solids on a linear path and
- Spinning – Extrusion of solids on a circular path.

HyperMesh also supports the creation of primitive solids through the

Geometry>Create>Primitives Pull Down. Supported primitives include:



Solid Editing

Tools for editing geometry:

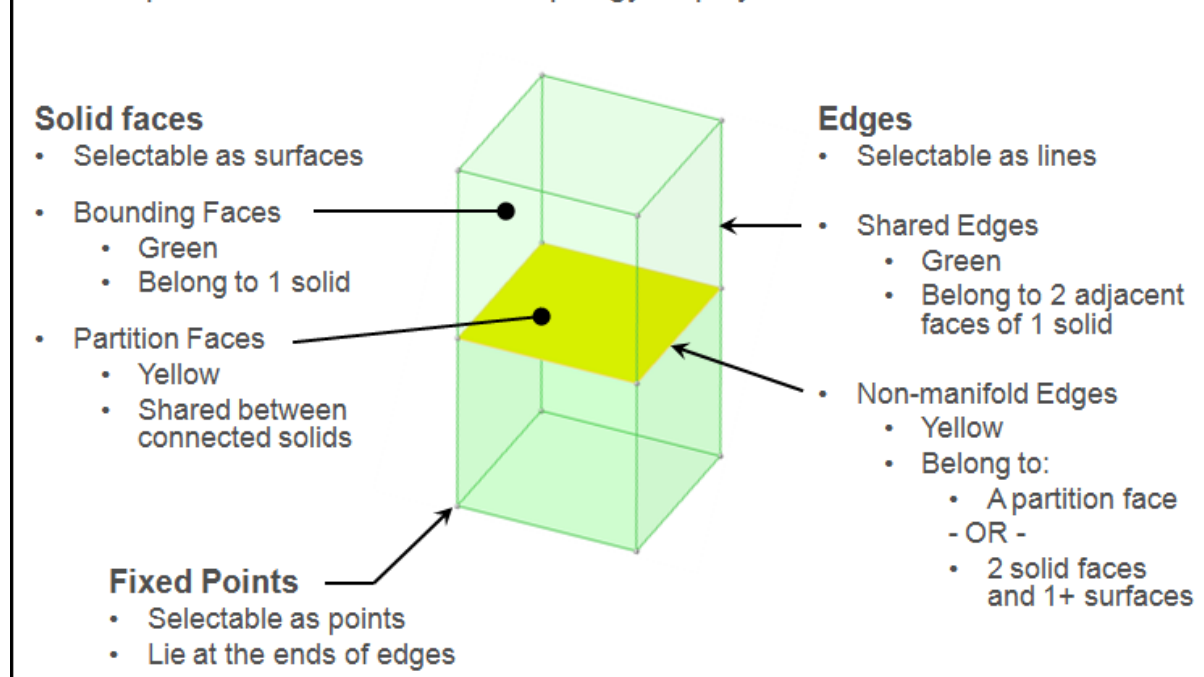
- **Surfaces** panel
 - Creates surfaces using various methods
 - Can be used to split a solid
 - Edges of the surface must be equivalenced to edges on the solid
- **Solid Edit** panel
 - **Trim with...** – splits a solid into 2+ solids using:
 - Nodes
 - Lines
 - Planes
 - Surfaces
 - **Merge** – combine 2+ adjacent solids into a single solid entity
 - **Detach** – disconnects connected solids
 - **Boolean** – advanced trim & merge operations
 - **Union** ($Solid A + Solid B$) – same as **merge**
 - **Intersection** ($Solid A \times Solid B$) – keeps overlapping portions of 2 solids
 - **Removal** ($Solid A - Solid B$) – deletes the volume of one solid from another
 - **Cut** ($Cut Solid A with Solid B$) – trims one solid with another solid

- keeps both solids, but they no longer overlap

Solid Topology

Topology governs the connectivity of solids and thus the connectivity of the elements created from them. Similar to 2D topology, 3D topology is represented by colors as follows:

- Example: 2 connected solids in topology display

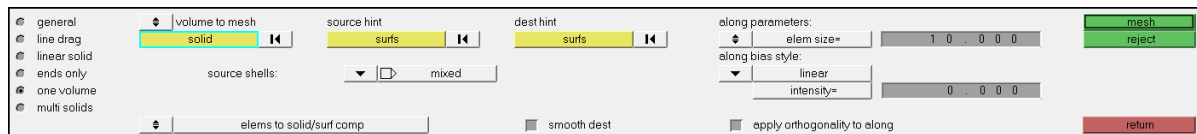


3D Topology visualization can be controlled using the **Visualization Browser**



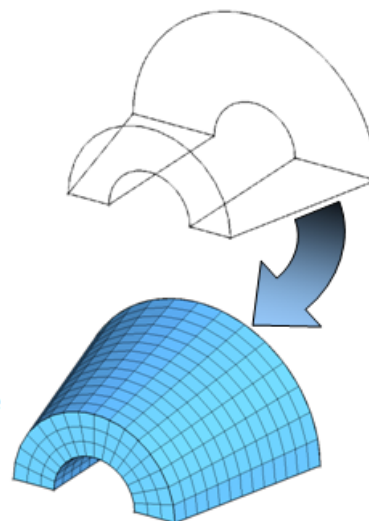
Solid Meshing

To mesh solid geometry, the Solid Map panel is used.



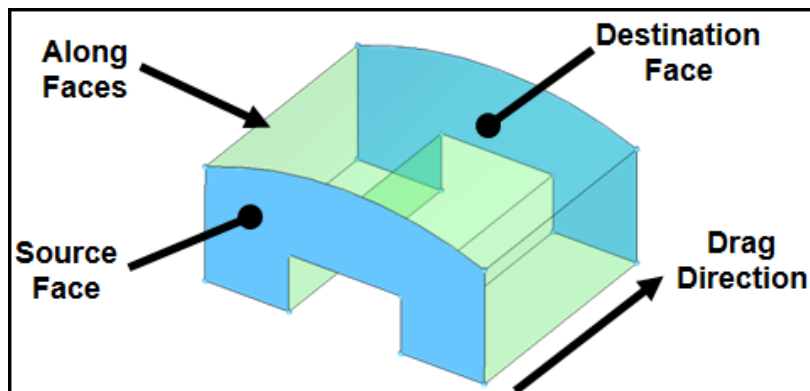
In particular, the one volume and multi solid sub panels. These allow for the automatic creation of Hexa/Penta mesh on mappable shapes.

- Location:
 - 3D (page) > **solid map** (panel)
 - OR -
 - **Mesh** (drop down menu) > **create** > **Solid Map**
- What it does:
 - Creates hexa-penta mesh in 1 or more volumes
 - Each volume is defined by selecting a solid geometry entity
 - Easy to define shape for the mesh since only one entity is selected
 - Each volume must be a “mappable shape”



Mappable Shapes

To use the automatic **Solid Map** function the geometry must be split into mappable shapes. Mappable shapes are defined as 2 opposing faces (source and destination) and faces that directly connect the source and destination (along faces).



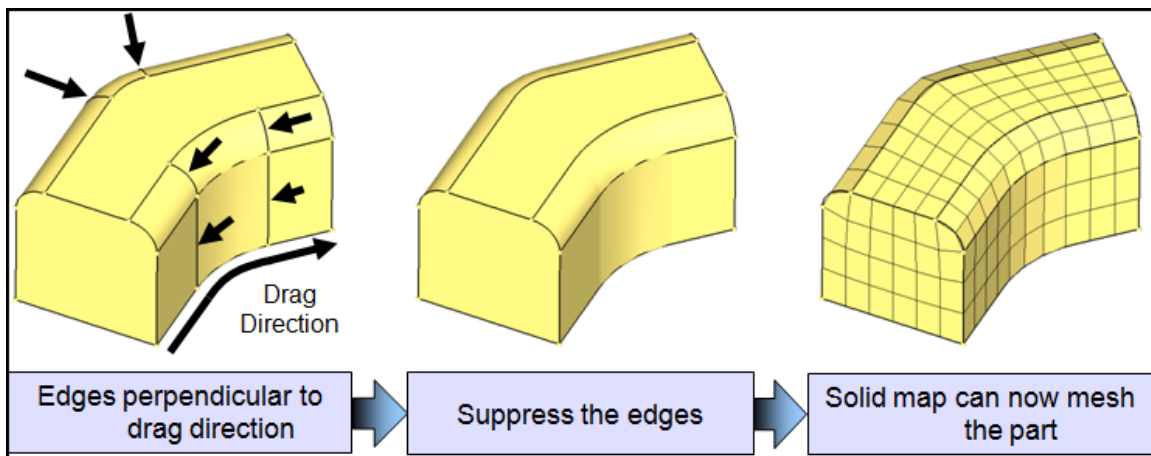
While this example shows two faces that are of the same shape and directly oppose each other, that is not a requirement. The source and destination can be of drastically different shape and contour and need not lie directly opposite each other.

Requirements and Tips

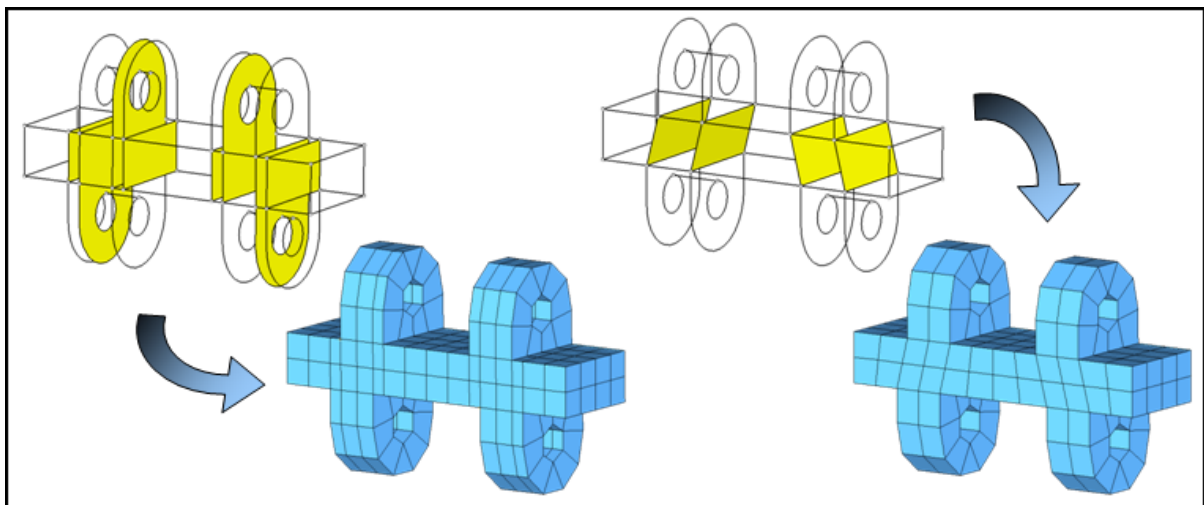
- Shape must be a closed volume
- Multiple source faces are allowed
- Destination face must be a single face

Requirements and Tips cont.

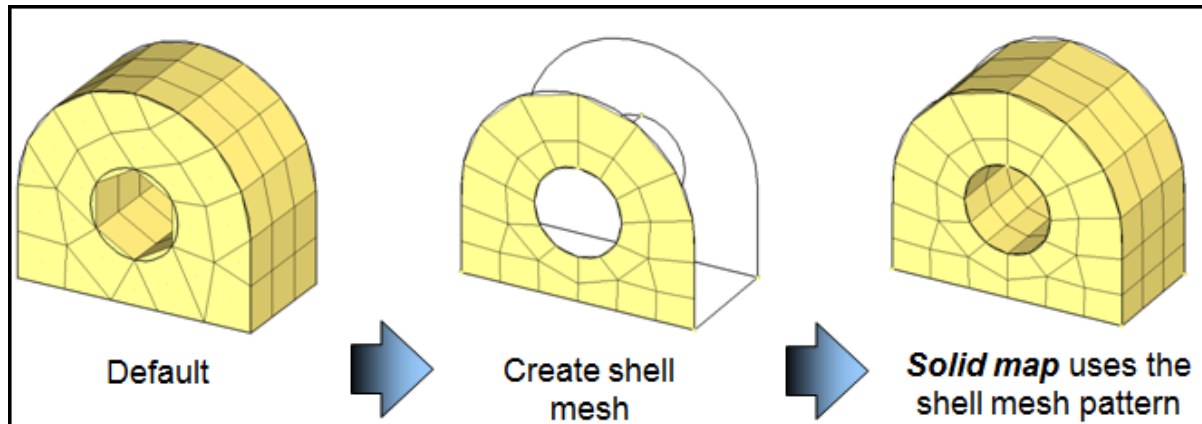
- No edges are permitted that are perpendicular to the drag direction. If they exist they must be suppressed.



- When splitting solids into Mappable Shapes it is recommended to split the part into the fewest possible regions to save time and offer more control over the mesh size. This often means splitting parts in ways that are not perpendicular to surfaces.

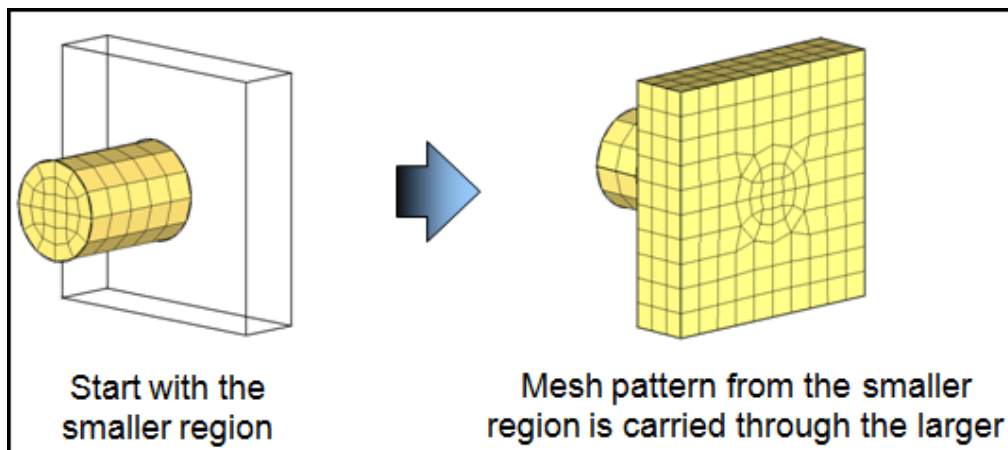


- Placing a 2D mesh onto a “Source” face of a mappable shape will allow control over the pattern of the resulting 3D mesh.

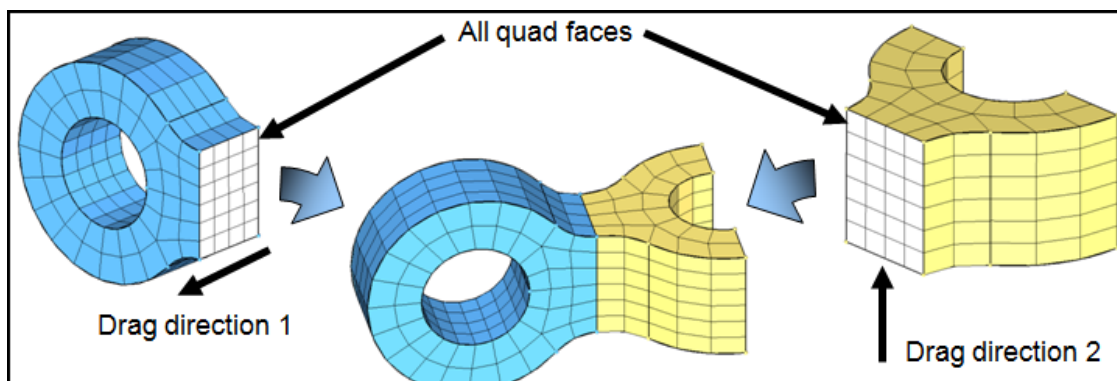


Requirements and Tips cont.

- When splitting solids into mappable regions, shared faces will guarantee 3D element connectivity across the split. This connectivity will also assure that the mesh pattern is carried through the part as subsequent regions are meshed.

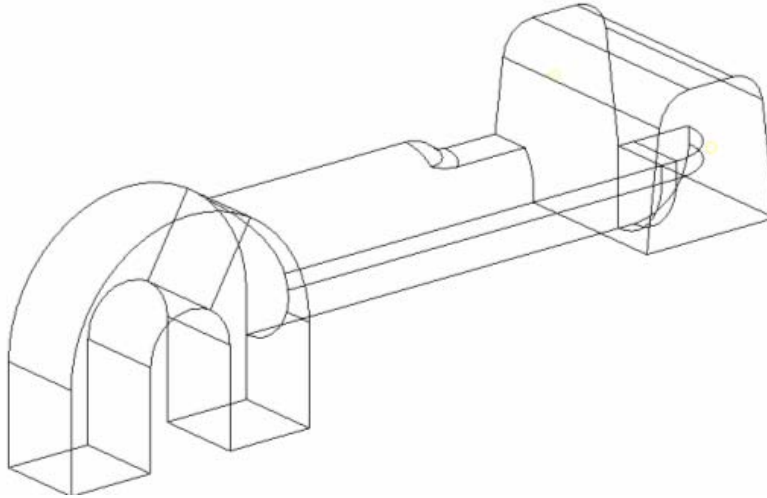


- Mesh connectivity of properly split regions with shared faces is also guaranteed even when mesh directions of the individual regions are not the same. **NOTE**: The mesh pattern on the “along” faces will always be quads.



Exercise: Creating and editing solid geometry

This exercise uses the model file, `solid_geom.hm`.



Step 1: Retrieve model file, `solid_geom.hm`

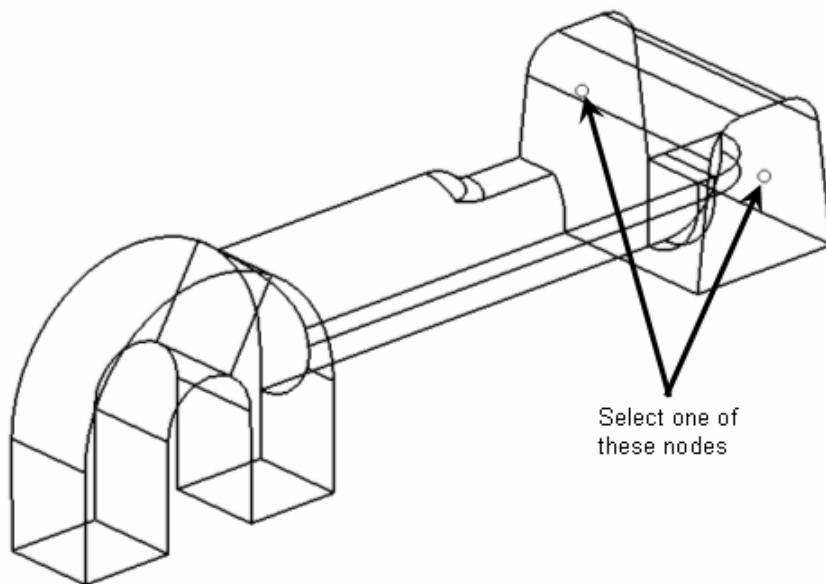
Step 2: Create solid geometry from the bounding surfaces.

1. From the **Geom** page, enter the **solids** panel.
2. Verify that you are in the **bounding surfs** sub-panel.
3. Verify that the **auto select solid surfaces** option is checked.
4. Select one surface on the part.
5. All of the surfaces should automatically be selected.
6. Click **create** to create the solid.
7. The **message** bar indicates that a solid has been created. The solids are identified by thicker lines than surfaces.
8. Click **return** to return to the main menu.

Step 3: Create a solid geometry cylinder using primitives.

9. Go to the **primitives** panel in one of the following ways:
 - From the **Geometry** pull-down menu, point to **Create**, and click **Primitives**
 - From the **Geom** page, go to **primitives**
10. Go to the **cylinder/cone** sub-panel.
11. Toggle **full cone** to **full cylinder**
12. Click **bottom center**, and select one of the temporary nodes (see following image).

13. The cursor automatically advances to the **normal vector** button.



14. Select the remaining temporary node shown in the image.

15. For **base radius=** enter 1 . 5 .

16. For **height=** enter 25.

17. Click **create solid**.

A solid cylinder is created in the middle of the first solid that was created.

18. Click **return** to return to the main menu.

Step 4: Subtract the cylinder's volume from the rest of the part.

1. Enter the **solid edit** panel in one of the following ways:

- From the **Geometry** pull-down menu, point to **Edit**, and click **Solids**
- From the **Geom** page, go to **solid edit**

2. Go to the **boolean** sub-panel.

3. Verify that **operation type:** is set to **simple (combine all)**.


4. Set **operation:** to **A-B (remove B from A)**.

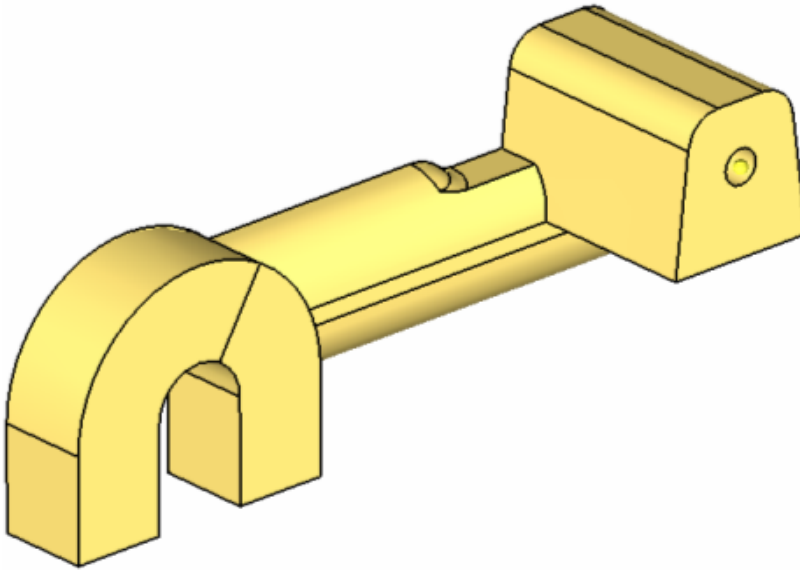
5. With the **solids** entity selector for **A:** active, select the original solid.

Verify that the cursor advances to **solids** next to **B:**.

6. Select the solid cylinder created in step 3.

7. Click **calculate**.

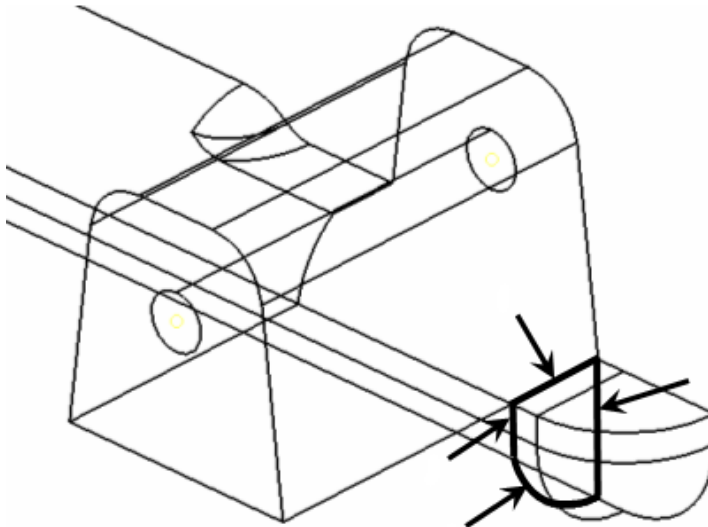
8. To confirm the material has been removed, click **shaded**, , and rotate the model to inspect the part.



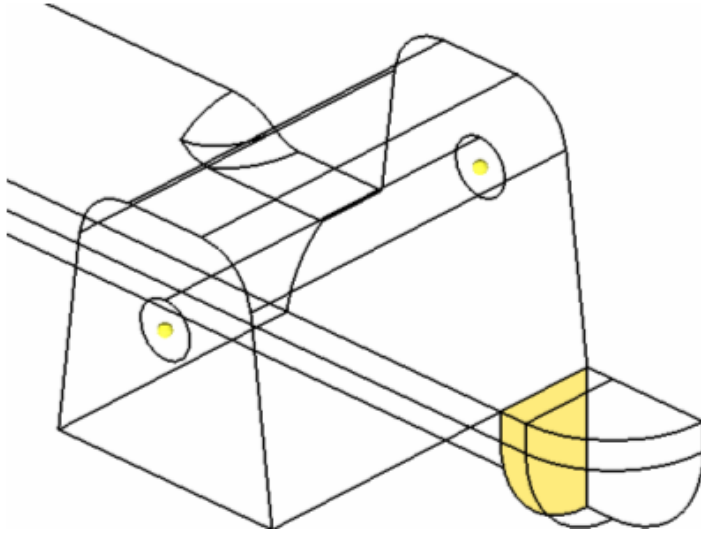
Step 5: Split the solid geometry using bounding lines.

You should still be in the **solid edit** panel.

1. Go to the **trim with lines** sub-panel.
2. Under with **bounding lines**:, activate the **solids entity** selector, and click anywhere on the model to select it.
3. Activate the **lines** entity selector and, in the Graphics area, select the lines shown in the following image.



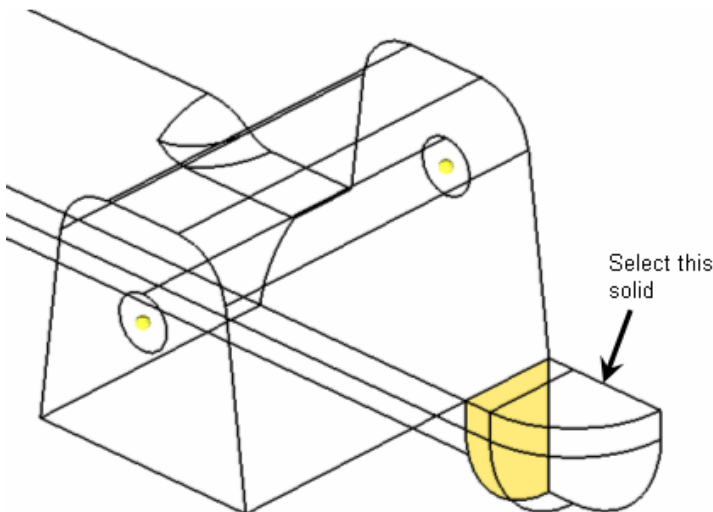
4. Click **trim** to trim the model.
A plane was trimmed and indicate that two solids now intersect.




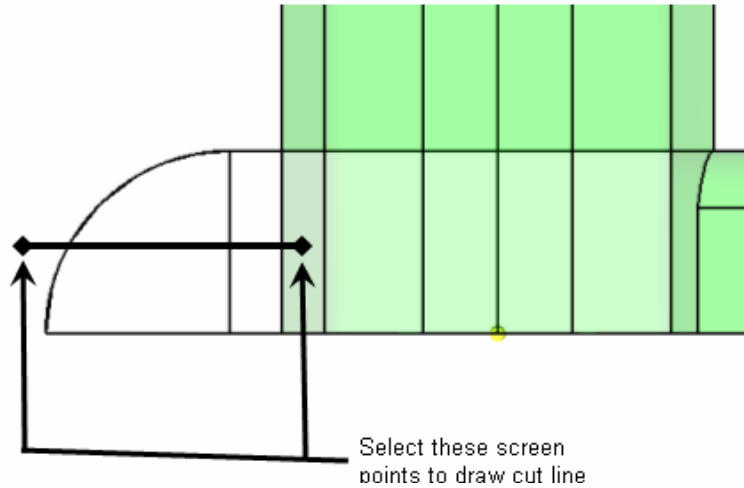
Step 6: Split the solid geometry using a cut line.

You should still be in the **solid edit** panel, **trim with lines** sub-panel.

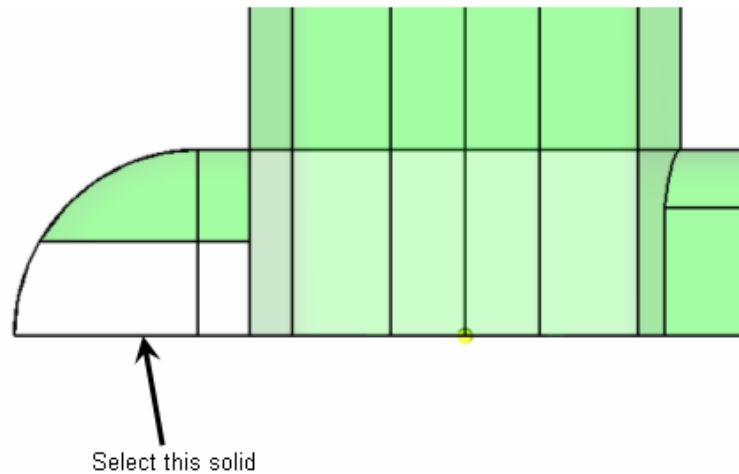
1. Under with **cut line**, activate the **solids** entity selector, and select the small, tetrahedral shaped solid created in step 5.



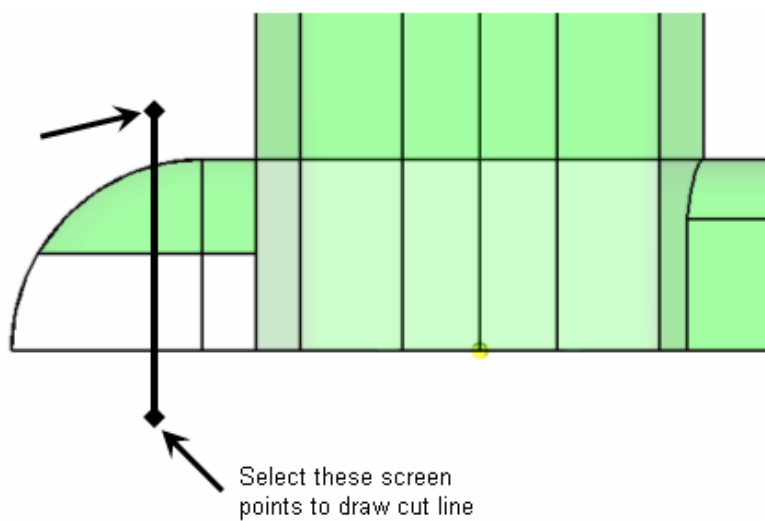
2. On the **toolbar**, click **User Views**, .
3. From the pop-up window, click **restore1**.
4. Click **drag a cut line**.
5. Pick two locations on screen such that they define the endpoints of a line that roughly divides the tetrahedral solid in half, as shown below.



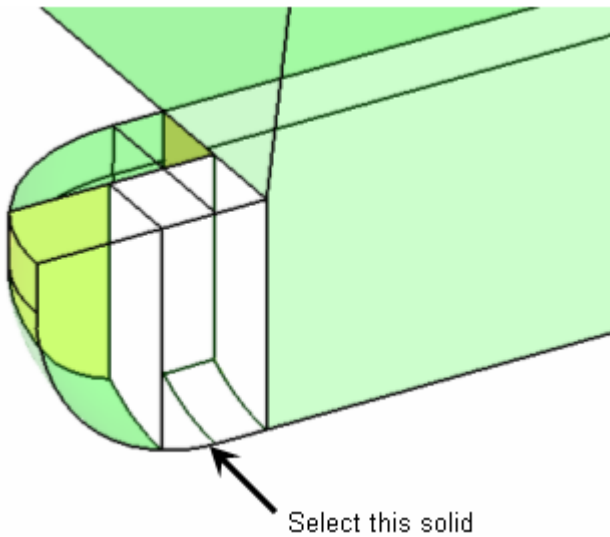
6. Click the middle-mouse button to split the solid.
7. Select the half of the original tetrahedral solid as shown in the following image.



8. Use **with cut line**: to split the solid as shown in the following image.

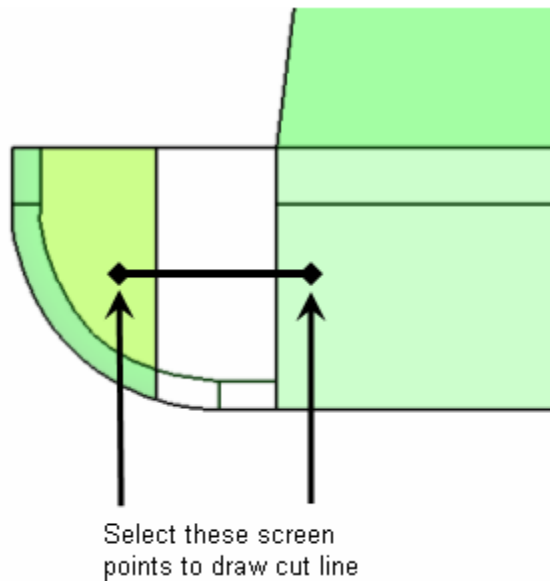


9. Select the solid shown in the following image.



10. Click the **User Views** icon, , and select **restore2**.

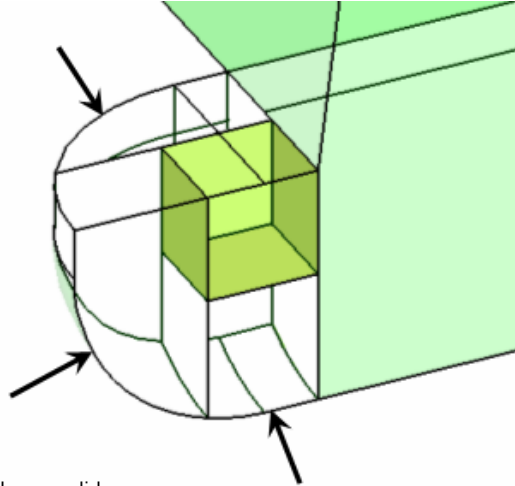
11. Use with **cut line**: to split the solid as shown in the following image.



Step 7: Merge solids together.

You should still be in the **solid edit** panel.

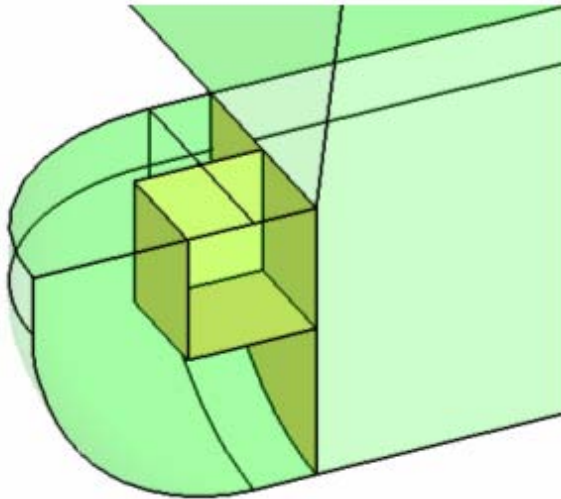
1. Go to the **merge** sub-panel.
2. With the **solids** entity selector under **to be merged**: active, select the three solids shown in the following image.



Select these solids


3. Click **merge** to merge the solids.

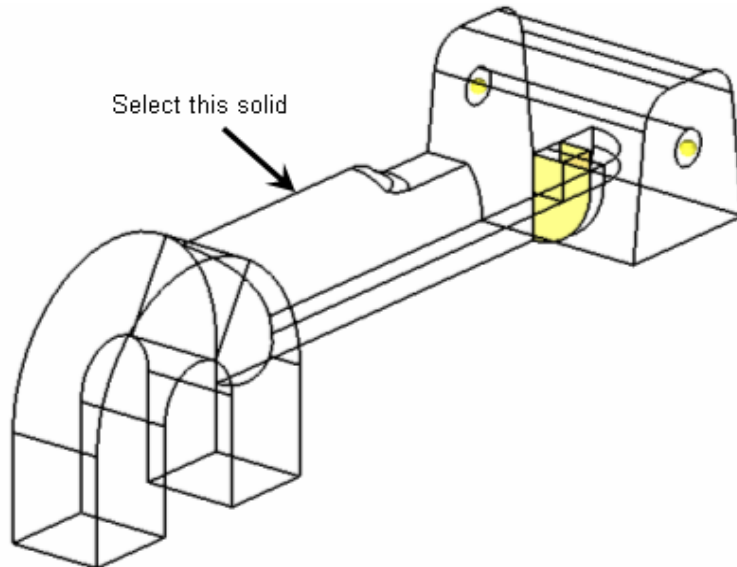
The resulting solids in the tetrahedral area should look like the following image. There should be two solid entities, with one of them being hexahedral in shape in the corner.



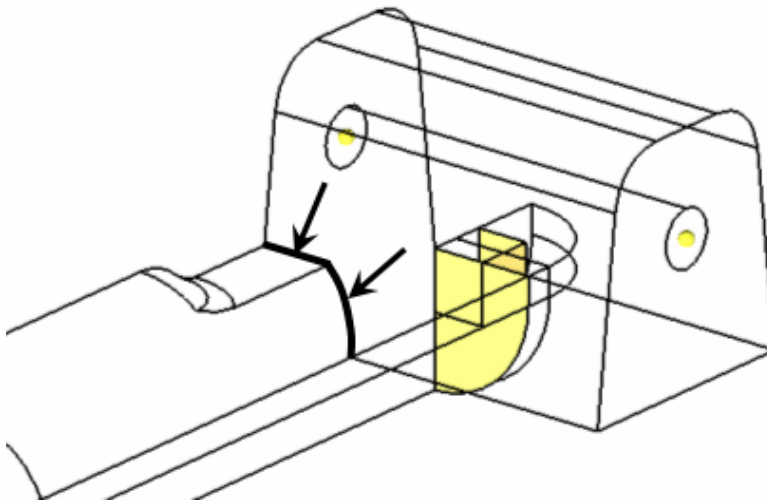
Step 8: Split the solid geometry with a user-defined plane.

You should still be in the **solid edit** panel.

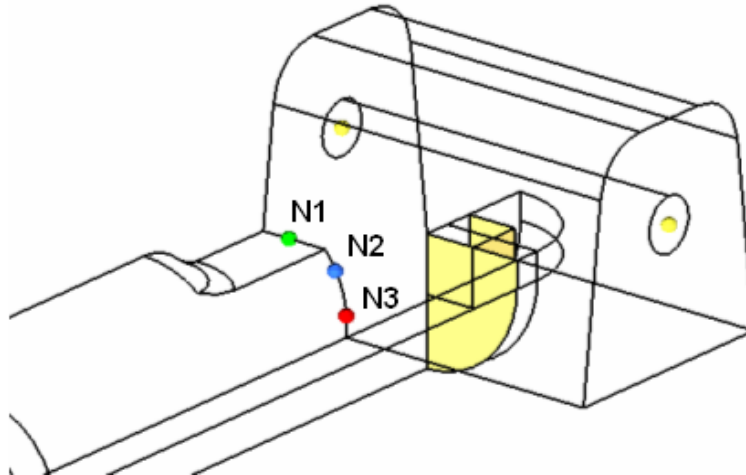
1. Go to the **trim with plane/surf** sub-panel.
2. Click the **User Views** icon, , and click **restore3**.
3. With the **solids** entity selector under **with plane:** active, select the large solid representing the majority of the part.



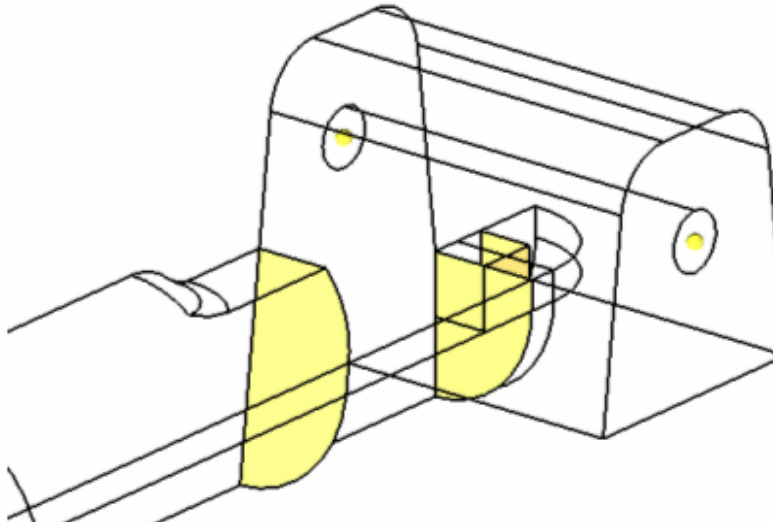
4. Set the **plane** selector to **N1, N2, N3**.
5. With N1 active, press and hold the left mouse button, and move the mouse cursor over one of the two edges shown in the following image.
The edge should highlight.



6. Release the mouse button, and left-click in the middle of the edge.
A green temp node appears at the location to indicate the selection for N1. The plane selector is advanced to the N2 selection.
7. In the same manner, highlight the other line shown in the image and select two nodes along its length.
Your selection should look similar to the following image.



8. Click **trim** to trim the solid.

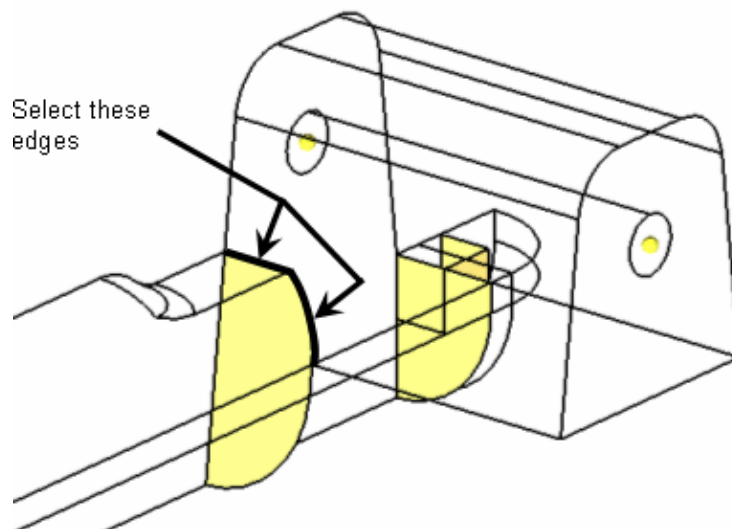


Step 9: Split the solid geometry with a swept line.

You should still be in the **solid edit** panel.

1. Go to the **trim with lines** sub-panel.
2. With the **solids** entity selector under with **sweep lines**: active, select the solid with the cylinder removed.

3. Activate the **line list** entity selector and select the edges shown below.



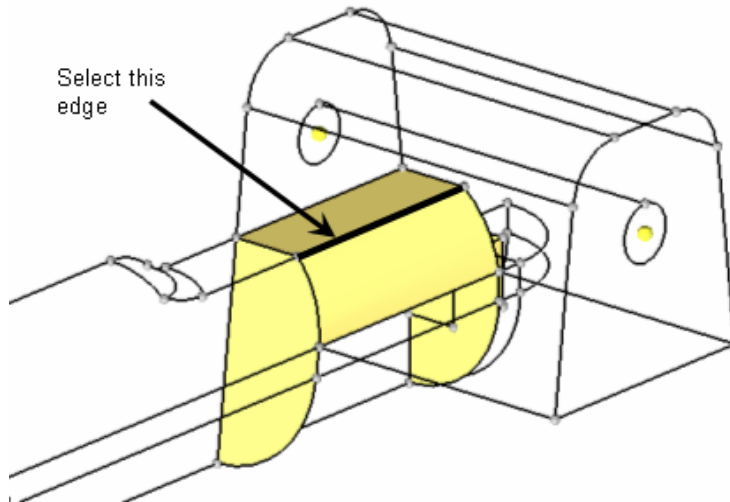
4. Under **sweep to:**, switch the plane selector from **N1, N2, N3** to **x-axis**.
5. Verify the panel is set to **sweep all** below the **plane** selector.
6. Click **trim** to trim the solid.

Step 10: Split the solid geometry with a principal plane.

You should still be in the **solid edit** panel.

1. Go to the **trim with plane/surf** sub-panel.
2. With the **solids entity** selector under with plane: active, select the solid with the cylinder removed.
3. Switch the **plane** selector from **N1, N2, N3** to **z-axis**.
4. Press and hold the left mouse button, and move the mouse cursor over the edge shown in the following image.

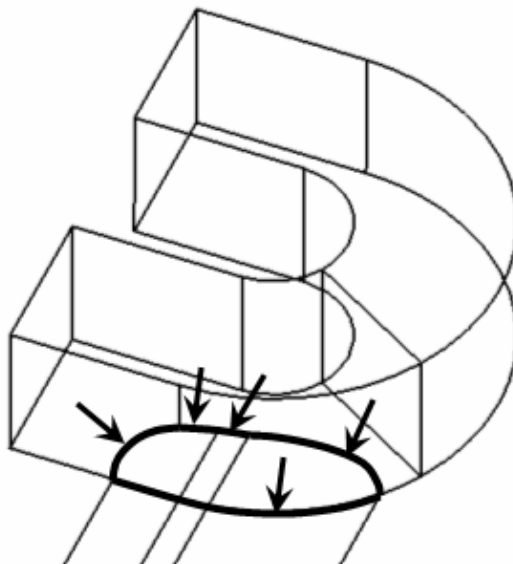
The edge should highlight.



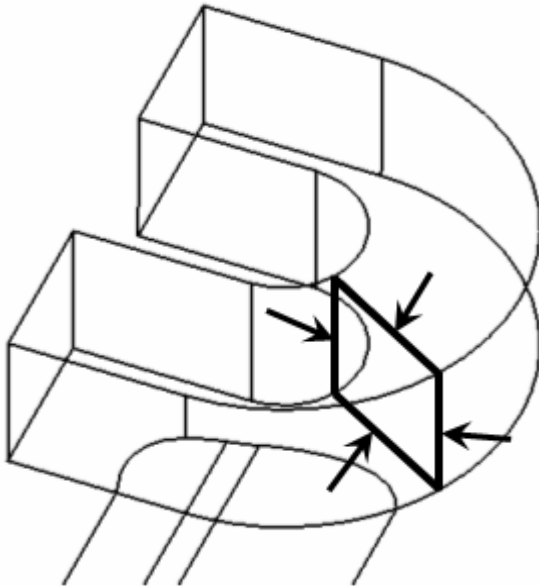
5. Release the mouse button, and left-click anywhere along the edge.
6. A purple temp node appears at the location to indicate the selection for the base node.
7. Click **trim** to trim the solid.
8. Click **return** to return to the main menu.

Step 11: Split the solid geometry by creating surfaces inside the solids.

1. Enter the **surfaces** panel in one of the following ways:
 - From the **Geometry** pull-down menu, point to **Create**, and click **Surfaces**
 - From the **Geom** page, go to **surfaces**
2. Go to the **spline/filler** sub-panel.
3. Deactivate **auto create (free edge only)** and **keep tangency** options.
4. Select the five lines shown in the following image:



5. Click **create** to create the surface.
6. Click **return** to return to the main menu.
7. From the **Geom** page, enter the **solid edit** panel.
8. Go to the **trim with plane/surf** sub-panel.
9. Under **with surfs:**, with the **solid** entity selector active, select the solid in the **Graphics** area.
10. Under **with surfs:**, with the **surfs** entity selector active, select the surface in the Graphics area that was just created.
11. Click **trim**.
12. Click **return**.
13. From the **Geom** page, go to the **surfaces** panel.
14. Go to the **spline/filler** sub-panel.
15. Select the four lines shown in the following image.

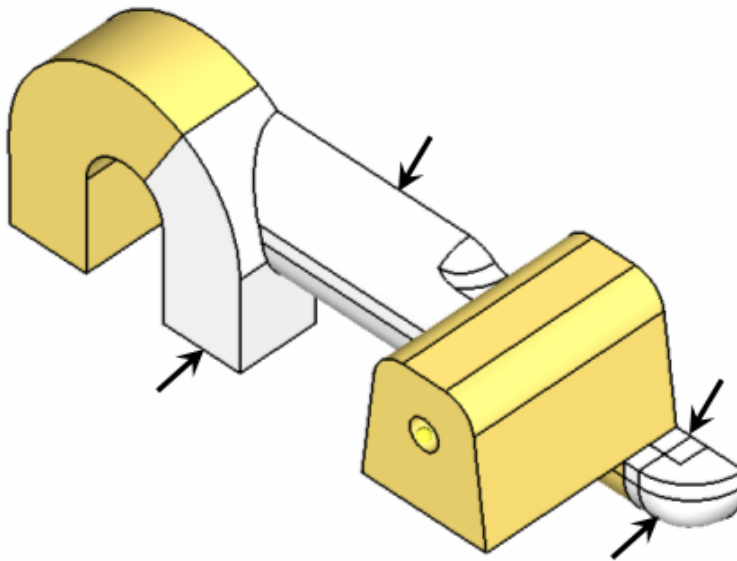


16. Click **create**.
17. Click **return**.
18. Go to the **solid edit** panel in one of the following ways:
 - From the **Geometry** pull-down menu, point to **Edit**, and click **Solids**
 - From the **Geom** page, go to **solid edit**
19. Select the **trim with plane/surf** sub-panel.
20. Under **with surfs:** activate **solids** and click the solid in the **Graphics** area.
21. Under **with surfs:**, with the **surfs** entity selector active, select the surface in the Graphics area that was just created.
22. Uncheck the **Extend Trimmer** box.

23. Click **trim**.
24. Click **return** to return to the main menu.

Step 12: Suppress extraneous edges on the part.

1. Enter the **edge edit** panel.
2. Go to the **(un)suppress** sub-panel.
3. Select **lines >> by geoms**.
4. With the **solids** entity selector active, select the four solids shown in the image below.



5. Click **add to selection**.
6. Set **breakangle =** to 45.
7. Click **suppress** to suppress the edges.
8. Click **return** to return to the main menu.

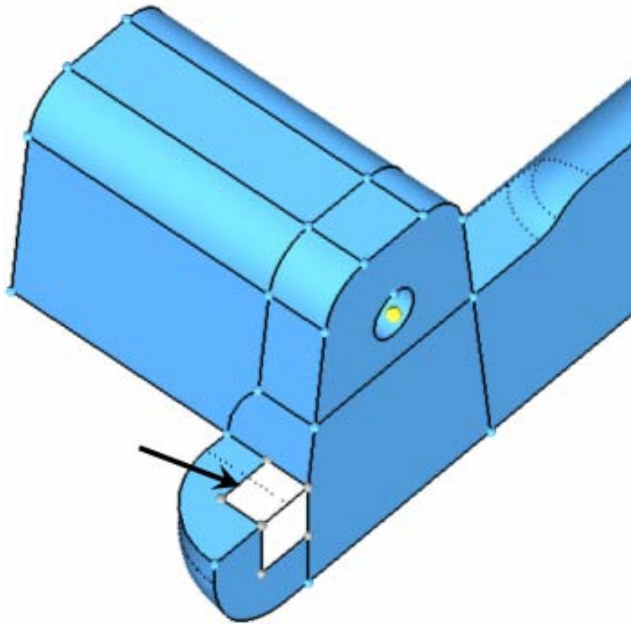
Hex-meshing the split Solid Geometry


Step 13: While you can use the model you created in the previous exercise, it is recommended you retrieve model file, **solid_map.hm**.

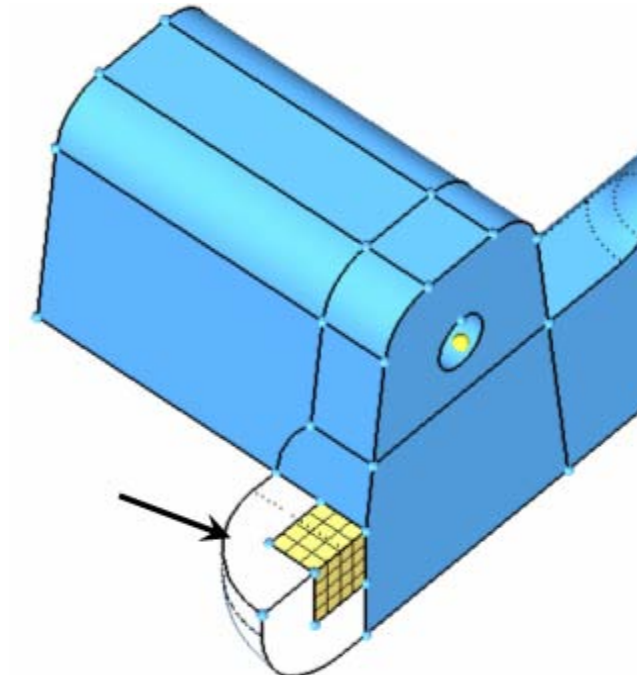
Step 14: Mesh the 1/8th sphere-shaped region.

1. On the **toolbar**, click **Shaded Geometry & Surfaces** (.
2. Enter the **solid map** panel.

- From the **Mesh** pull-down menu, point to **Create**, and click **Solid Map Mesh**
 - From the **3D** page, go to **solid map**
3. Go to the **one volume** sub-panel.
 4. Under **along parameters**:, set **elem size** to 1.
 5. With the **solid** entity selector under **volume to mesh** active, select the small cube-shaped solid, as shown in the following image.



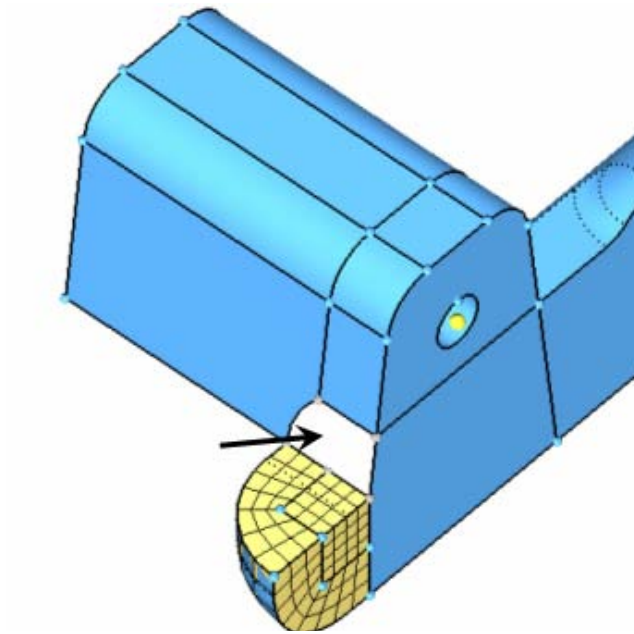
6. Click **mesh**.
7. On the **toolbar**, click **Shaded Elements & Meshlines** icon ().
8. Select the solid indicated in the following image.



9. Click **mesh**.
10. Click **return** to return to the main menu.

Step 15: Create a shell mesh with the automesh panel to control a mesh pattern.

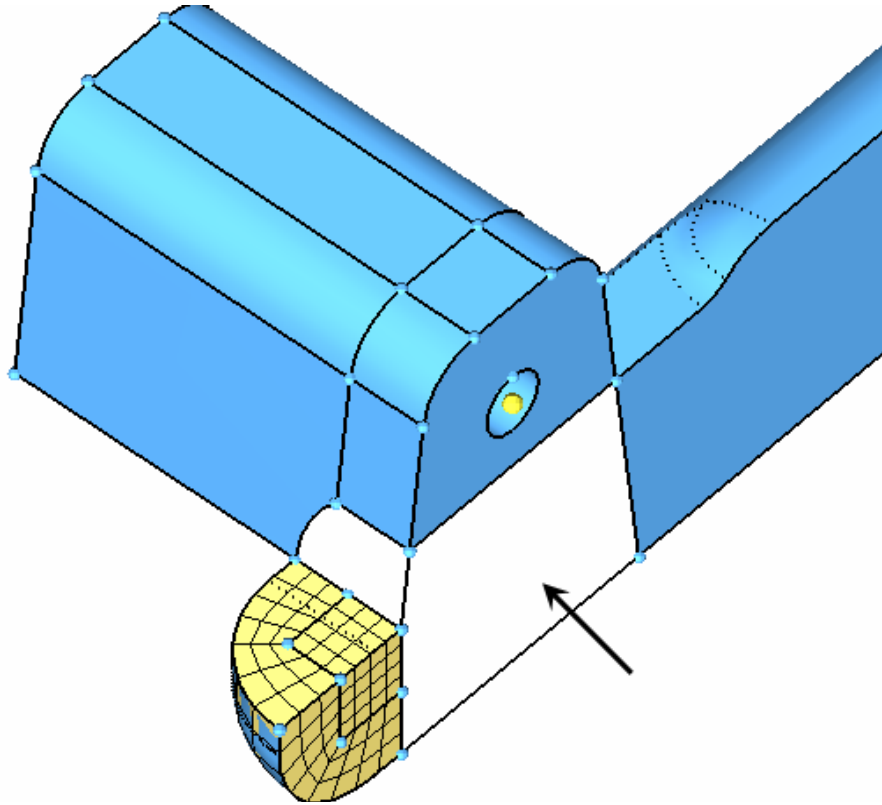
1. Go to the **2D** page.
2. Enter the **automesh** panel in one of the following ways:
 - From the **Mesh** pull-down menu, select **Create**, and click **2-D AutoMesh**
 - Press F12
3. Select the surface indicated in the following image.




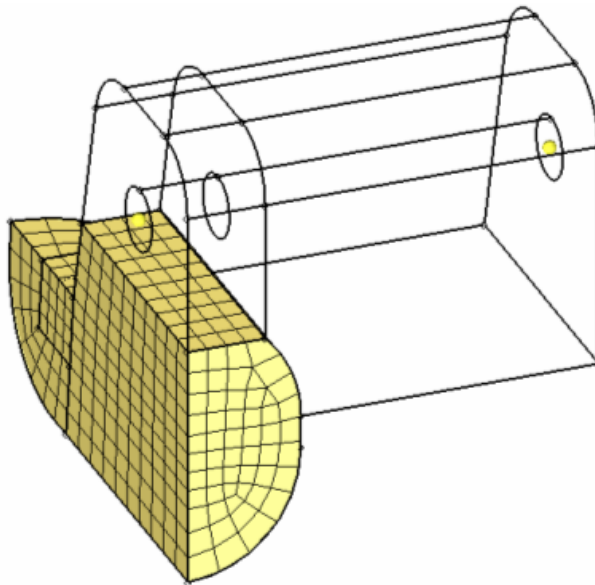
4. Verify that the meshing method is set to **size and bias** and **interactive**.
5. Verify that **element size =** is set to 1.000.
6. Verify that **mesh type:** is set to **mixed**.
7. Click **mesh**.
8. In the **elem density =** field, enter 4.
9. Click **set all to**.
All the densities are set to 4.
10. Click **mesh**.
11. Click **return** to return to the main menu.

Step 16: Mesh the solid volume on which the surface mesh was created in step 15.

1. Go to the **3D** page.
2. Enter the **solid map** panel.
3. Go to the **one volume** sub-panel.
4. Select the volume shown in the following image.



5. Under along parameters:, toggle **elem size=** to **density=** and enter 10.
6. Click **mesh**.
7. On the **toolbar**, click the **Wireframe Geometry** icon ().
8. Rotate the part and note how the mesh pattern created with the automesh panel has been used for the solid elements.



Step 17: Mesh the remaining solid volumes.

You should still be in the ***solid map*** panel.

1. Select one of the remaining unmeshed solid volumes.

Make sure to select a solid adjacent to one that has already been meshed so that connectivity is maintained.

2. Switch ***source type:*** to ***mixed***.
3. Under ***along parameters:***, toggle ***density=*** to ***elem size=*** and enter 1 . 5.
4. Click ***mesh***.
5. Repeat until all solid volumes are meshed.
6. Click ***return*** to return to the main menu.

Step 18 (Optional): Save your work.

With this section of the exercise completed, you can save the model if desired.

Automated Solid Map Meshing




The capability to automate the solid map meshing process is now available. Using the “Mappable” visualization mode in conjunction with the multi-solids feature will inform you that the solid(s) are ready for solid meshing. Using the multi-solids feature will allow for all solids within the model to be meshed in one step provided that they are mappable.

In this section of the tutorial we will delete all the elements from the previous section, use the Mappable visualization mode and solid mesh the part using multi-solids.

Step 19: Delete the elements within the model.

1. Press F2 to access the ***delete*** panel.
2. Select all the elements.
3. Click ***delete***.
4. Click ***return*** to return to the main menu.

Step 20: Use the Mappable visualization mode.

1. On the ***toolbar***, click ***Shaded Geometry & Surfaces*** ().
2. From the ***geometry visualization*** pull-down, select ***Mappable*** ( Mappable ).

You will notice that each solid is color coded, the color of the solid represents its mappable state. The goal is to ensure that each solid is either 1-directional or 3-directional mappable.

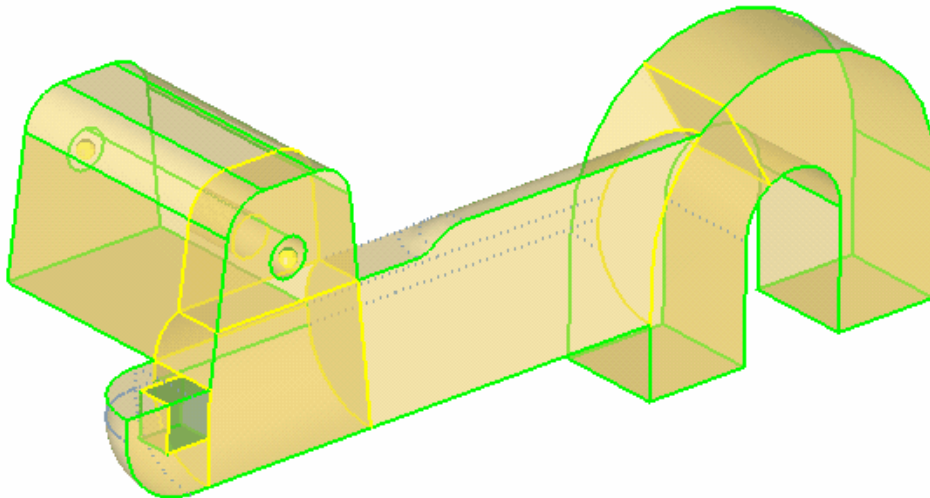
The color of the mappable state can be found (and adjusted if necessary) within the ***Options>>Color*** panel.

ignored map:	■ Color
not mappable:	■ Color
1 dir. map:	■ Color
3 dir. map:	■ Color

By mappable display control (solids)

- ***ignored map*** Default visualization for solids that require partitioning to become mappable.
- ***not mappable*** Visualization for solids that have been edited, but still require further partitioning to create mappable solids.
- ***1 dir. map*** Visualization for solids that can be mapped (for 3-D meshing) in one direction.
- ***3 dir. map*** Visualization for solids that can be mapped (for 3-D meshing) in three directions.

Once in the mappable visualization mode it is clear that there is one 3-directional mappable solid, the rest are 1-directional mappable.

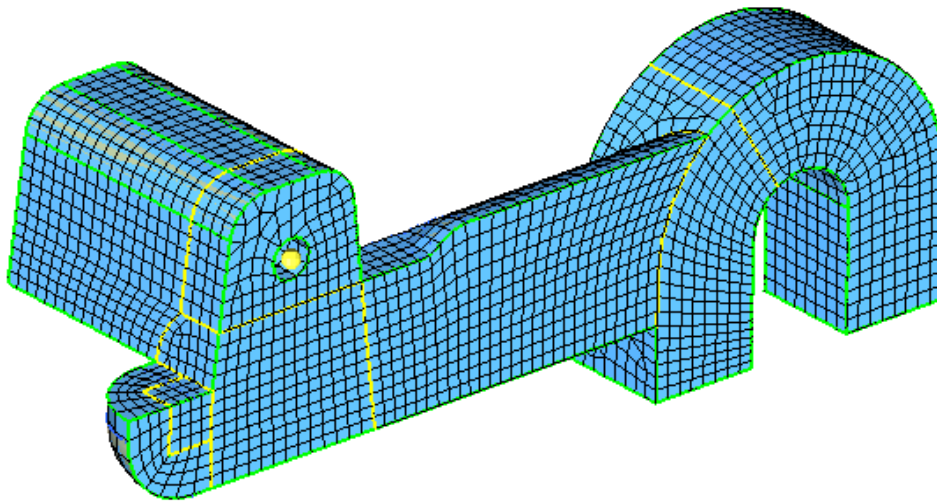


Step 21: Use the Multi-solid feature to mesh the part.

1. From the main menu, go to the **3D** page.
2. Enter the ***solid map*** panel.

3. Go to the **multi solids** sub-panel.
4. Select **solids>>all**.
5. Under **source shell type**, select **mixed**.
6. Set **elem size** to 1.
7. Click **Mesh**.

The solids will be sequentially solid meshed. Once complete inspect the model and note that all the mesh within the solids is correctly equivalenced.



Step 22 (Optional): Save your work.

With this exercise completed, you can save the model if desired.

Chapter 5

Tetra Meshing

HyperMesh provides several methods of generating a tetrahedral mesh. The standard method creates tetras from an enclosed volume of shell elements, plus several parameters. This provides the user with a lot of control over the final tetra mesh. The volume tetra mesher quickly and automatically creates a tetrahedral mesh on an enclosed volume of surfaces or solid geometry with only a few inputs. Finally, the quick tetra mesher creates a tetra mesh that maintains user specified quality requirements, but may sacrifice details in the shape of the part to do so. All methods are valid in certain situations. The exercise in this section focuses on the standard and volume tetra meshing methods.

In this chapter, you will learn about:

- Volume tetra mesher
- Standard tetra mesher
- Checking tetra element quality
- Re-meshing tetra elements

Standard Tetra Meshing

Standard Tet Meshing involves wrapping a volume in 2D elements and then using that predefined mesh to “grow” Tetra elements to fill the volume. The process is as follows;

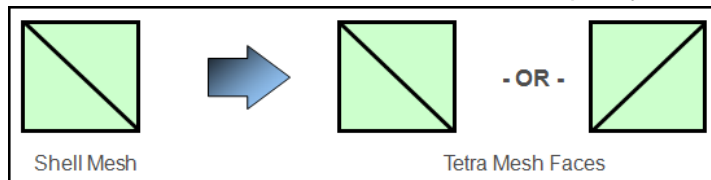
1. Generate a surface mesh of shell elements
2. Check quality and connectivity of the plate elements
3. Generate the tetrahedral mesh
4. Delete the original surface mesh
5. Edit if necessary to obtain good quality

The standard method of Tetra Meshing can be found through the Mesh>Create>Tetramesh Pull down and then select the *tetra mesh* subpanel.

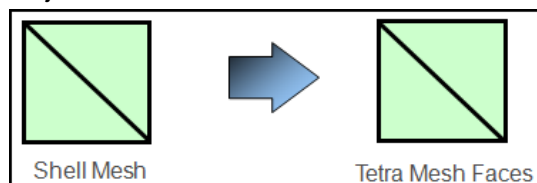
- **Requirements for the shell mesh:**
 - Enclose one, and only one, continuous volume
 - There can be no free or T-connected edges.
 - There can be no duplicates in the mesh.
 - Elements should not fold over and overlap each other
 - Avoid very low minimum tria angles
 - Avoid a large difference in size between adjacent elements
 - Avoid a large difference in size between two sides of a wall thickness
 - **For quad elements in the shell mesh:**
 - Split quads into 2 trias and create tetra elements under them
- OR –
- Keep the quad element and create pyramids under them

When using the Standard Tetramesh, the user will select the trias/quads that will define the mesh and optionally the user can select fixed trias/quads. HyperMesh will, when creating the tetra mesh, flip the diagonal of the 2D elements if it deems the resulting tetra mesh will be of a higher quality with the flip. HyperMesh will not do this to elements selected in the fixed selection.

- **Standard Trias/Quads**
 - Adjacent tria faces on the tetrahedral mesh may have their diagonal reversed from the shell mesh if tetras are better quality

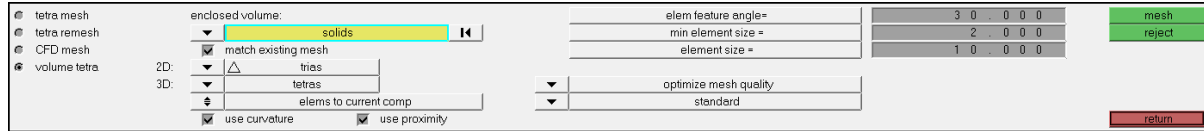


- **Fixed Trias/Quads:**
 - Adjacent tria faces on the tetrahedral mesh always match the shell mesh



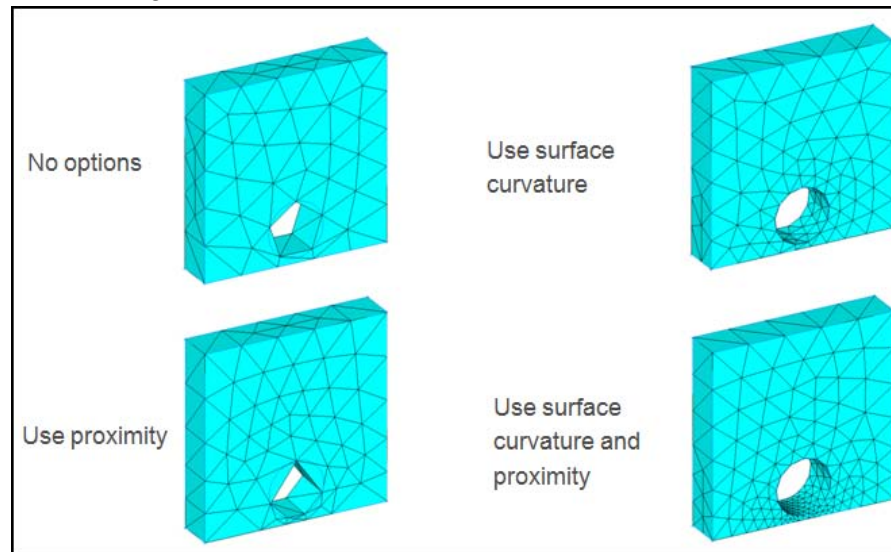
Volume Tetra Meshing

The volume tetra meshing utility, found in the Mesh>Create>Tetramesh Pull down and in the *volume tetra* subpanel, provides a quick method for generating tetra mesh. No initial 2D mesh is required and the tetra mesh can be generated on solid geometry or inside surfaces fully bounding a volume.



Two options are available to control the mesh;

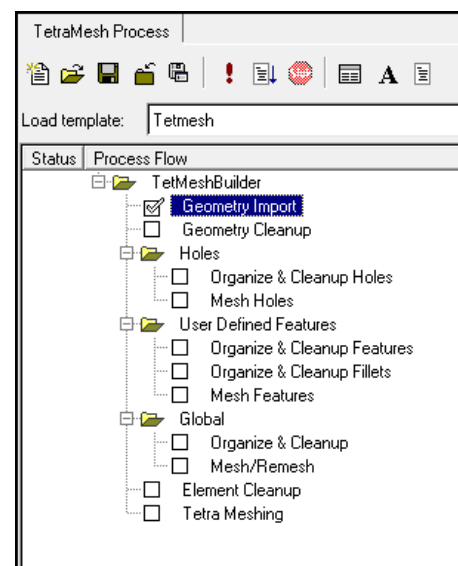
- **Use Proximity** – Creates smaller elements next to small features to make a smooth transition from small to large elements.
- **Use Curvature** – Will place more elements along curved surfaces based on user specified settings.



TetraMesh Process Manager

The final, and most powerful method for creating a tetra mesh is the TetraMesh Process manager found in the Mesh>Create>TetMesh>Create New Pull down. This uses the incorporated process manager to provide a step by step process to guide the user through import, cleanup and organization of the geometry, establishment of individual mesh pattern, and creation and cleanup of the tetra mesh.

The user can control organization, mesh style and ultimate mesh quality. More information can be found in the online help and tutorials.



Exercise: Tetramesh a Housing

Step 1: Retrieve and view the model file.

The model for this exercise is `housing.hm`. Take a few moments to observe the model using the different visual options available in HyperMesh (rotation, zooming, etc.).

Only the geometry in the component **cover** is currently displayed. The file contains two parts defined by a volume of surfaces. The geometry has been cleaned such that surface connectivity is proper and surface edges that would cause sliver elements are suppressed.

Step 2: Use the volume tetra mesher and equilateral triangles to create a tetra mesh for the cover.

1. Access the tetramesh feature in one of the following ways:

- From the **Mesh** menu, point to **Create**, and click **Tetra Mesh**
- On the **3D** page, click the **tetramesh** sub-panel

2. Go to the **volume tetra** sub-panel.

3. Set the **entity** selector to **surfs** and select one of the surfaces in the model.

The connected surfaces are selected automatically.


4. Verify that **2D:** is set to **trias** and **3D:** is set to **tetras**.

These control the type of element that will be created for the surface mesh and solid mesh of the part.

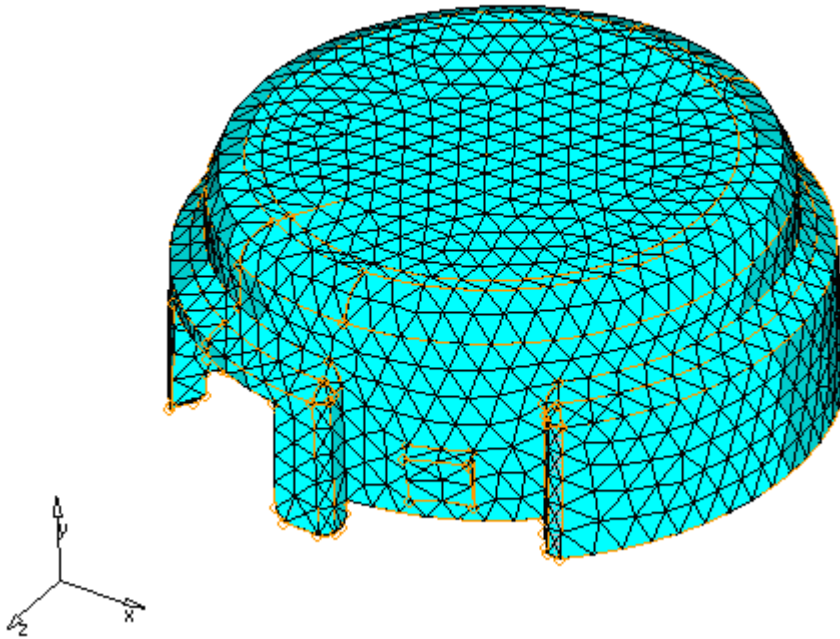
5. Verify that the **use curvature** and **use proximity** options are off.

6. For **element size=** specify 10.

7. Click **mesh** to create the tetra mesh.

8. From the **toolbar**, click the **shaded elements** icon, .

9. Take a moment to inspect the mesh pattern that the volume tetra mesher created.



Tetra mesh from the **volume tetra** sub-panel and equilateral triangles (**2D: trias**)

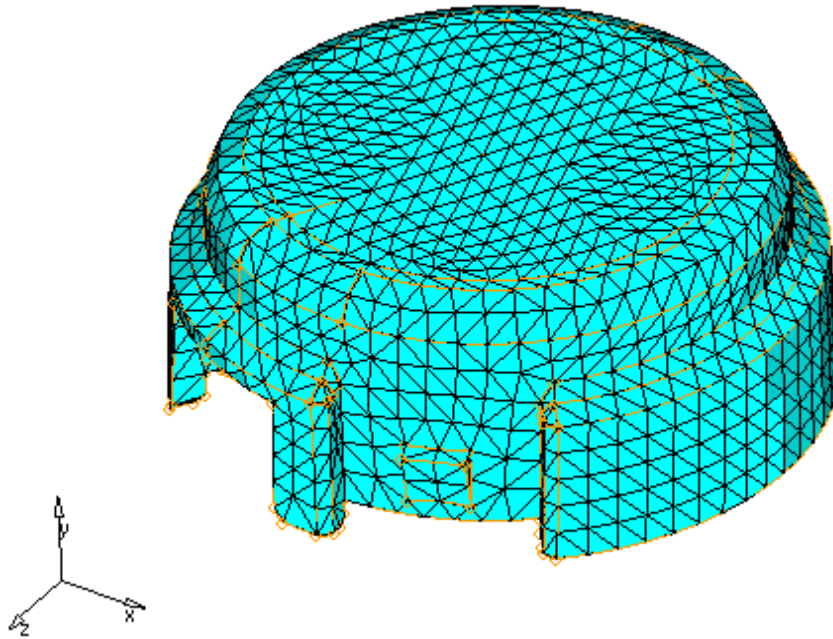
10. Click **reject** to reject the mesh.

The mesh is deleted.

Step 3: Use the volume tetra mesher and right triangles to create a tetra mesh for the cover.

You should still be in the **tetramesh / volume tetra** sub-panel.

1. Select one of the surfaces in the model.
2. Switch **2D:** to **R-trias**.
3. Click **mesh** to create the tetra mesh.
4. Inspect the mesh pattern that the volume tetra mesher created.
5. Compare it to the first mesh you created and note the differences. The 2D: R-trias setting tends to create tetra elements with triangular faces that are right triangles (90-45-45 angles) while the 2D: trias setting tends to create equilateral triangles (60-60-60 angles).



Tetra mesh from the **volume tetra** sub-panel and right triangles (**2D: R-trias**)

6. **Reject** the mesh.

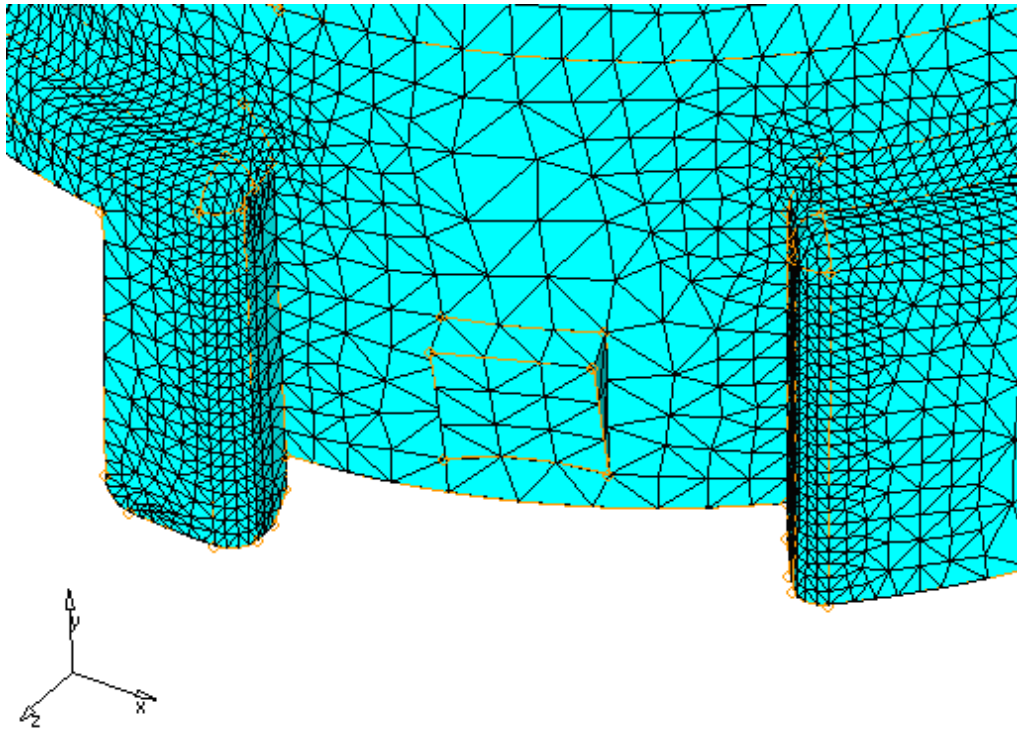
Step 4: Use the volume tetra mesher to create a tetra mesh with more elements along curved surfaces.

You should still be in the **volume tetra** sub-panel.

1. Select one of the surfaces in the model.
2. Activate the option, **use curvature**.

Additional parameters appear. The option, use curvature, causes more elements to be created along areas of high surface curvature. Thus, curved areas such as fillets will have more and smaller elements, which capture those features with higher resolution.

3. Verify that **elem feature ang =** is set to 30.
4. For **min element size =** specify 1 . 0.
5. Click **mesh** to create the tetra mesh.
6. Shade the tetrahedral elements.
7. Inspect the mesh pattern that the volume tetra mesher created.
8. Compare it to the previous meshes you created and note the differences. More elements are created around the fillets.



Tetra mesh from the **volume tetra** sub-panel and the option **use curvature** active

9. Click **reject** to reject the mesh.

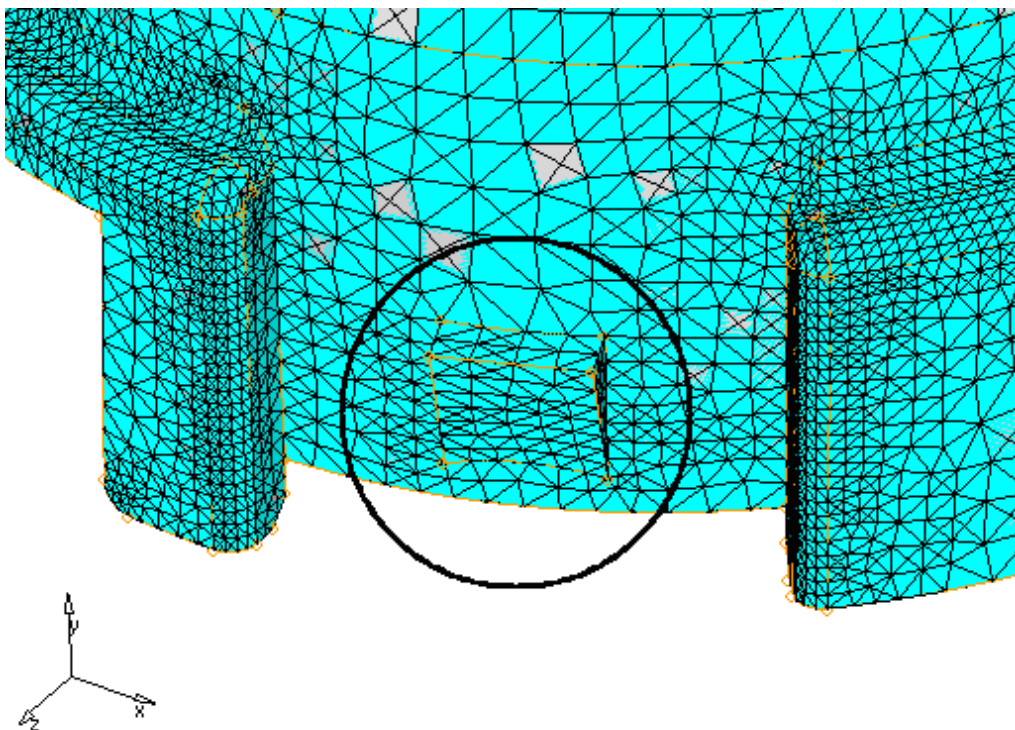
Step 5: Use the volume tetra mesher to create a tetra mesh with more elements around small features.

You should still be in the **volume tetra** sub-panel.

1. Select one of the surfaces in the model.
2. Activate the **use proximity** option.

The use proximity option causes the mesh to be refined in areas where surfaces are smaller. The result is a nice transition from small elements on small surfaces to larger elements on larger, adjacent surfaces.

3. Click **mesh** to create the tetra mesh.
4. Shade the elements.
5. Inspect the mesh pattern that the volume tetra mesher created. Compare it to the previous meshes you created and note the differences. More elements were created around surfaces with small angles as indicated in the following image.



Tetra mesh from the volume tetra sub-panel with options, *use curvature* and *use proximity*

6. Click **return** to exit to the main menu.

Step 6: Prepare the display to tetra mesh the hub component using the standard tetra mesher.

1. Use the **model browser** to turn off the display for all components for geometry.
2. Turn on the display of the geometry for the **hub** component.
3. Use the **model browser** to turn off the display for all components for elements.
4. Turn on the display for the **hub** and **tetras** components for elements.

There are tria shell elements in the hub component. Currently, there are no elements in the tetras component.

Step 7 (Optional): Review the connectivity and quality of the tria mesh to validate its integrity for the standard tetra mesher.

Use the edges and check elems panels to make sure that there are no free edges or very small angles in the tria shell mesh.

1. Access the **edges** panel in one of the following ways:
 - From the **Pull-down** menu, select **Mesh**, then **Check**, then **Components**, then **Edges**
 - From the main menu, select the **Tool** page and the **edges** panel.

2. With the **comps** selector active, pick any tria element on the **hub** component.
3. Click **find edges**.

A message in the header bar should state, "No edges found. Selected elements may enclose a volume."

This is desired as the tetra mesher requires a closed volume of shell elements.

4. Click **return** to exit to the main menu.
5. Access the **check elements** panel in one of the following ways:
 - From the **Pull-down** menu, select **Mesh**, then **Check**, then **Elements**, then **Check Elements**
 - From the main menu, select the **Tool** page and the **check elems** panel.
6. Verify that you are in the **2-d** sub-panel.
7. Identify elements having an **aspect** ratio greater than 5.

Aspect ratio is the ratio of the longest edge of an element to its shortest edge. This check helps you to identify sliver elements.

All of the hub's shell elements pass the check; all of the elements have an aspect ratio less than 5.

8. Identify tria elements having an angle less than 20. (**trias: min angle**)

This check also helps you to identify sliver elements.

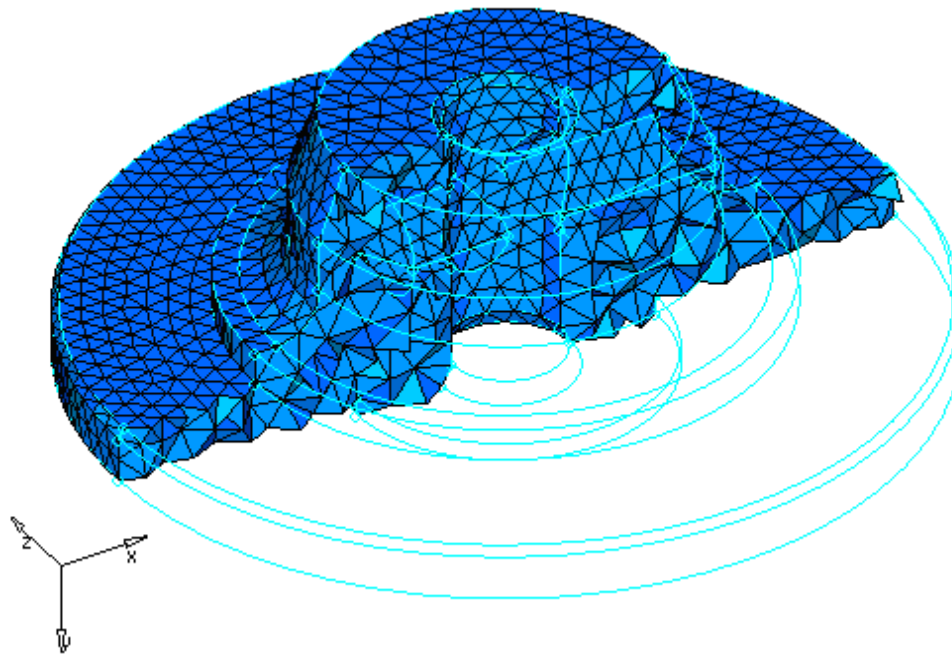
All the hub's shell elements pass the check; all the elements have angles greater than 20.

The surface mesh is suitable for creating a tetra mesh.

9. Click **return** to exit to the main menu.

Step 8: Create a tetra mesh for the hub using the standard tetra mesher.

1. Use the **model browser** to set the current component to **tetras**.
2. On the **3D** page, go to the **tetramesh** panel.
3. Access the **tetra mesh** sub-panel.
4. With the **comps** selector active under **select trias/quads to tetra mesh:**, select one of the hub shell elements from the Graphics area.
5. Elements or comps selected here give the tetra mesher the option to swap the diagonal for any pair of surface trias, if that will result in a better tetra mesh quality. If you do not want to change the element connectivity on the surface mesh, use the fixed trias/quads selector.
6. Click **mesh** to generate the tetrahedral elements.



Cut-away view of tetrahedral elements

Step 9: Check the quality of the hub's tetra elements.

1. Use the **model browser** to display only the elements in the **tetras** component.
2. Access the **check elems** panel.
3. Go to the **3-d** sub-panel.
4. Identify the smallest element **length** among the displayed elements.

If the minimum length is acceptable for a target element size of 5.0, then no further action is necessary.

5. Identify the smallest angle (**tria faces: min angle**) among the displayed elements.

If the minimum tria face angle is no less than 10° , then the mesh quality should be acceptable.

6. Identify elements having a **tet collapse** smaller than 0.3.

The tet collapse criteria is a normalized volume check for tetrahedral elements. A value of one indicates a perfectly formed element with maximum possible volume. A value of zero indicates a completely collapsed element with no volume.

The **message** bar indicates that one element has a tetra collapse smaller than 0.3.


Step 10: Isolate the element with the tetra collapse smaller than 0.3 and find the elements surrounding it.

You should still be in the **check elements** panel.

1. With 0.3 still specified for **tet collapse**, click **tet collapse** again.
2. Click **save failed**.

The element that failed the tetra collapse check is saved in the user mark, and can be retrieved in any panel using the extended selection menu.

3. Click **return** to exit to the main menu.
4. Access the **mask** panel in one of the following ways:

- From the **display** toolbar, select **Mask** (.
- Press F5 on your keyboard.

5. Set the entity selector to **elems** and select **elems >> retrieve**.


The element that was saved in the **check elems** panel is retrieved.

6. Select **elems >> reverse**.

7. **Mask** the elements.

Only the one tetra element that failed the tetra collapse check should be displayed.

8. Click **return** to exit to the main menu.

9. On the **display** toolbar, click **unmask adjacent** (.

The layer of elements that is attached to the one displayed element is identified and displayed.

10. Click **unmask adjacent** () two more times.

The layer of elements that is attached to the displayed elements is identified and displayed. The functionality of unmask adjacent can be duplicated using the **find** panel, **find attached** sub-panel on the **Tool** page.

Step 11: Remesh the hub's displayed tetra elements to improve their tetra collapse.

1. Access the **tetramesh** panel.
2. Go to the **tetra remesh** sub-panel.
3. Select **elems >> displayed**.
4. Click **remesh** to regenerate this area of the mesh.

Note that the re-meshing operation works on only one group of elements (one volume) at a time.

5. Click **return** to exit to the main menu.

6. Access the **check elems** panel.
7. Click **tet collapse** to find out if the tetra collapse has improved for the displayed elements.

The message in the header bar should indicate that the minimum tetra collapse is larger than the value reported before the tetra elements were re-meshed.

8. Click **return** to exit to the main menu.

Step 12 (Optional): Save your work.

Now that the tetra mesh is complete, it is a good time to save the model.

Summary

A tetra mesh has been created for both parts in the file. Different procedures for tetra meshing were used. Either method can be used to mesh parts, depending on the needs of the analysis. Also, the tetra re-mesh function was used to show how to quickly fix the quality of tetra elements.

Chapter 6

Analysis Setup

Section 1: Setting up Loading Conditions

A finite element solver can solve for responses of parts to loading conditions placed on them. The loads can be in the form of any combination of boundary constraints, forces, pressures, temperatures, etc. This section focuses on defining the loading conditions on a model.

In this section, you will learn how to:

- Create constraints (RADIOSS SPC) on the channel's geometry lines
- Create a force (RADIOSS FORCE) on the bracket to simulate a pressing load on it
- Define a load step (RADIOSS SUBCASE)

Analysis Setup

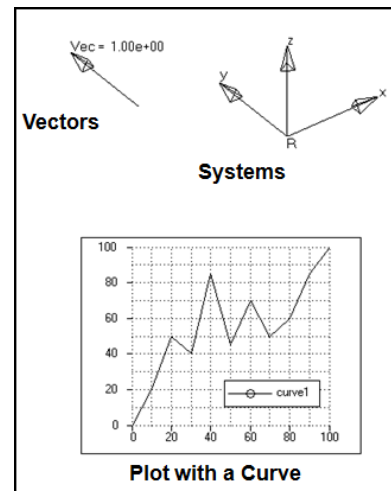
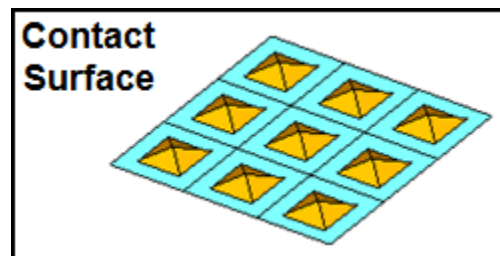
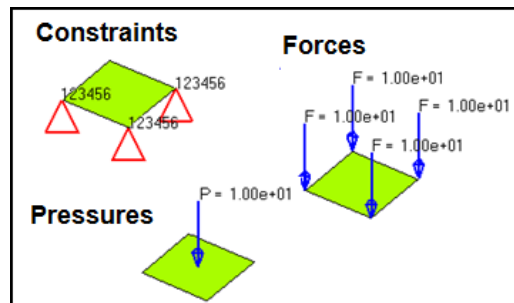
Analysis setup is the definition of all information for an analysis besides the mesh including:

- Specification of the solver to be used
- Creation materials, properties, etc.
- Assignment of a solver specific format to HyperMesh entities
- Creation boundary conditions (constraints, loads, contacts, etc.)
- Definition of other required information (solution requests, general run parameters, etc.)

This section will focus on the creation of boundary conditions

Supported Entities

- **FE Loading**
 - Loads (constraint, force, pressure, moment, temperature, flux, velocity, acceleration)
 - Equations (mathematical link between nodes)
 - Contacts
 - Group (defines contact between entities)
 - Contact Surfs (defines a list of entities that can be used as master or slave in a group)
- **Reference Entities**
 - Sets (a simple list of a particular type of entity)
 - Blocks (a list of entities contained within a box shape)
- **Coordinate Entities**
 - Systems (coordinate axes)
 - Vectors
- **Plotting**
 - Curves (X-Y data)
 - Plots (a display of curves with axes)
- **Output Requests**
 - Loadsteps (combinations of load collectors)
 - Output Blocks (request output from an analysis for certain entities)
- **Control cards** (job-level, global parameters for the analysis)

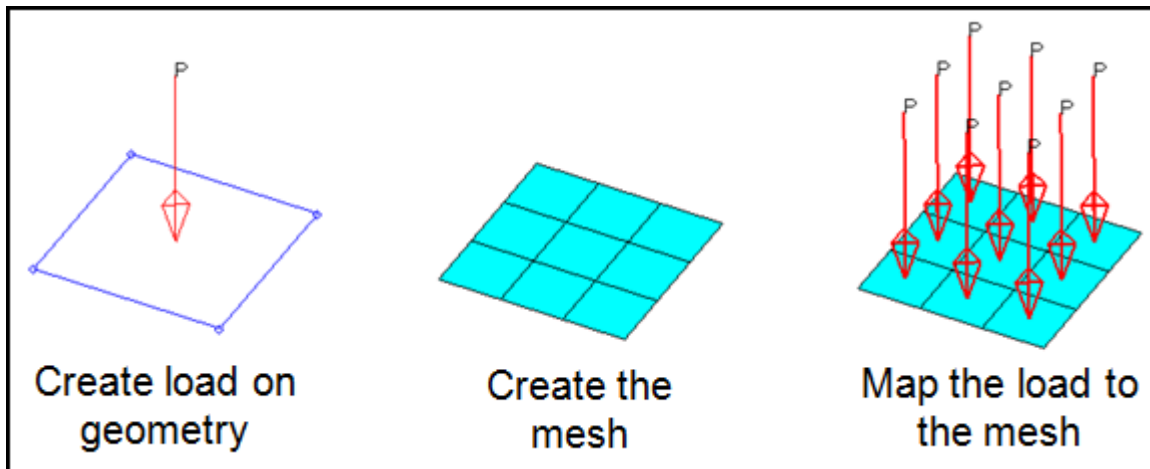
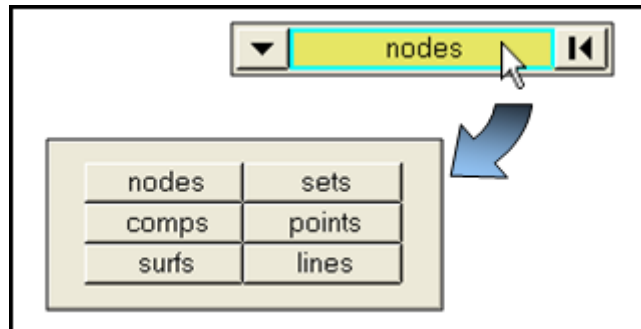


Loads on Geometry

HyperMesh allows the user to place loads on geometry before the mesh is ever created. Then using the **BCs>Loads on Geometry** pull down, those loads will be mapped to the elements that were created from that geometry. This is a time saving advantage as the user only needs to pick one, or at most a few, geometric entities to ultimately create hundreds or even thousands of loads or constraints.

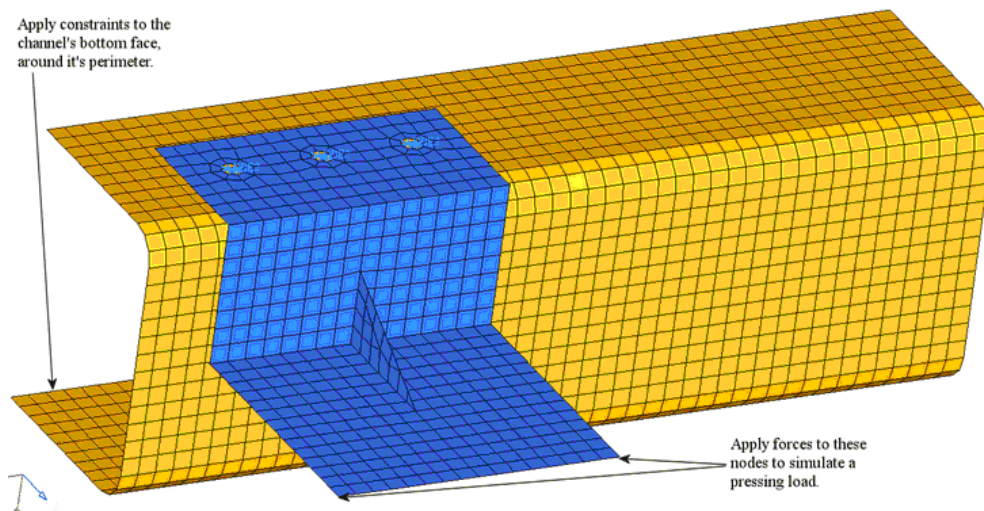
To use this, change the entity selector from a node or element selection to a geometric selection such as surfs or lines.

Then after the part is meshed, using the **Loads on Geometry** function, the loads will be automatically mapped to elements created from that geometry.



Exercise: Setting up Loading Conditions


This exercise uses the model file, `channel_brkt_assem_loading.hm`. It contains the bracket and channel assembly in the following image.




Step 1: Retrieve and view the HyperMesh model file and set profile.


1. Load the model `channel_brkt_assem_loading.hm`.
2. Set the user profile to **Radioss Bulk Data**

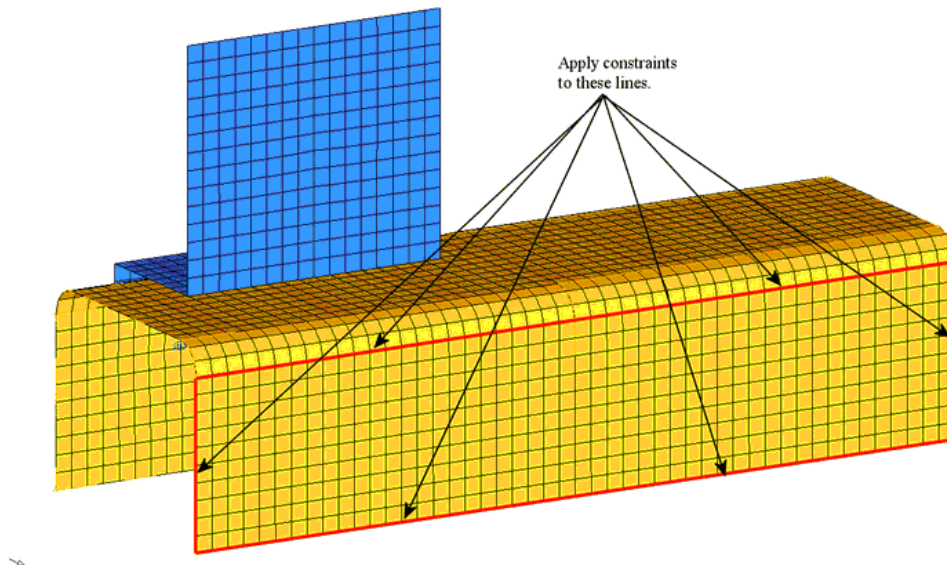
Step 2: Create two load collectors named *pressing_load* and *constraints*.

1. Access the **collectors** panel by clicking **load collectors** () on the **collectors** toolbar.
2. Go to the **create** sub-panel.
3. For **loadcol name** = enter `pressing_load`.
4. Select your desired color.
5. Switch the **creation method** to **no card image**.
6. **Create** the load collector ***pressing_load***.
7. For **loadcol name** = enter `constraints`.
8. Select your desired color.
9. **Create** the load collector ***constraints***.
10. **Return** to the main menu.

Step 3: Apply constraints (RADIOSS SPC) to the channel's line geometry.

1. Use the **model browser** to display the **geom** () for the component **channel**.

2. On the toolbar, click **User Views** (), and pick **iso1**.
3. Enter the **constraints** panel in one of the following ways:
 - From the **Pull-down** menu, select **BCs**, then **Create**, then **Constraints**
 - On the main menu, select the **Analysis** page, then select **constraints**
4. Go to the **create** sub-panel.
5. Switch the **entity** selector to **lines**.
6. Select the six lines on the perimeter of the channel's bottom surface as shown in the following image.




7. Activate degrees of freedom (dof) 1 through 6.
 For a RADIOSS linear static analysis, dof 1, 2, and 3 represent translations in the global x, y, and z directions respectively. Dof 4, 5, and 6 represent rotations about the global x, y and z axis, respectively.
8. Set **Load Types** = to **SPC**.
9. **Create** the constraints on the lines.
10. For **size** = enter 5.
 The display size of the constraints is reduced.
11. Activate the option **label constraints**.
 A label is displayed for each constraint. The labels identify which dofs are assigned to the constraints.
12. Click **return** to exit to the main menu.

Step 4: Map the constraints (RADIOSS SPC) on the geometry lines to the channel nodes associated to the lines.

1. Access the **load on geom** panel in one of the following ways:
 - From the **Pull-down** menu, select **BCs**, then **load on geom**
 - On the main menu, select the **Analysis** page, then select **Load on Geometry**
2. Select **loadcols >> constraints**.
3. Click **select** to complete the selection of load collectors.
4. Click **map loads**.

A constraint is at each node associated to the geometry lines.
5. Click **return** to exit to the main menu.
6. On the **model browser**, turn off the display of geometry for all component collectors.

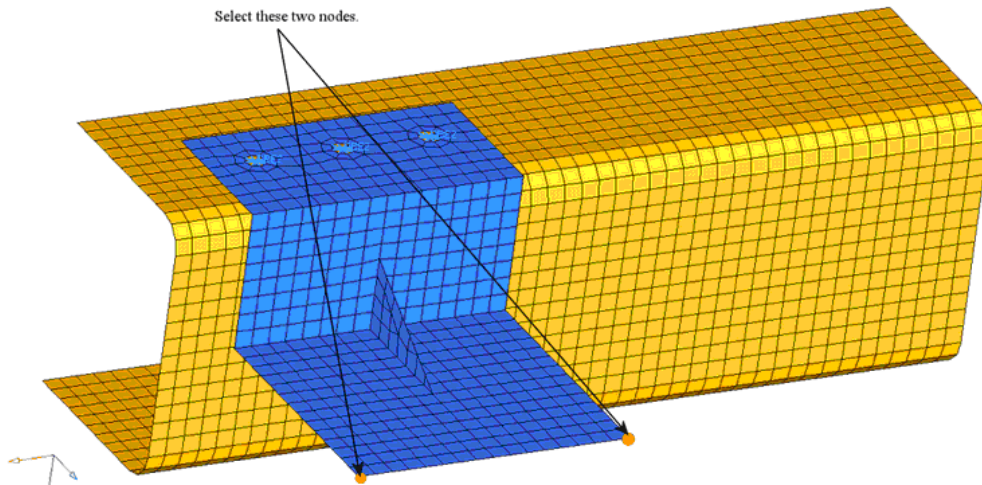
Step 5: Prepare to create forces (RADIOSS FORCE) on the bracket for the pressing load case.

1. On the toolbar, click **User Views** () and pick **restore1**.
2. On the **model browser**, right-click on the **pressing_load** load collector and select **Make Current**.

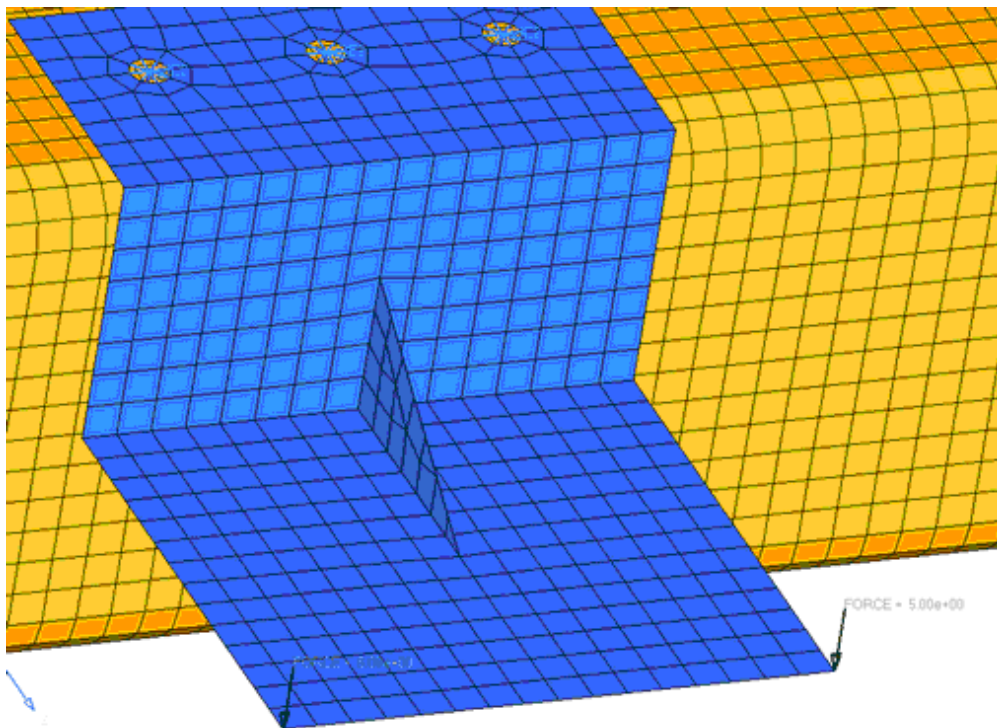
The pressing_load load collector is now the current load collector, and any loads created will be placed in this collector.

Step 6: Create two forces (RADIOSS FORCE) on the bracket for the pressing load case.

1. Access the **forces** panel in one of the following ways:
 - From the **Pull-down** menu, select **BCs**, then **forces**
 - On the main menu, select the **Analysis** page, then select **Forces**
2. Go to the **create** sub-panel.
3. With the **nodes** selector active, select the two nodes as indicated in the following image.



4. For **magnitude** = enter 5.
5. Switch the **direction** selector from **N1, N2, N3** to **y-axis**.
6. Set **Load Types** = to **Force**.
7. **Create** the forces.
8. For **magnitude %** = specify 200.0.
The display size of the force is increased.
9. Activate the option **label loads**.
Each force displays the label **FORCE = 5.00e+00**.



The two forces created for the pressing load case

10. Click **return** to exit to the main menu.

Step 7: Define the load step for the pressing load case.

1. Access the **load step** panel in one of the following ways:
 - From the **Pull-down** menu, select **Setup**, then **Create**, then **Load Steps**
 - On the main menu, select the **Analysis** page, then select **load steps**

2. For **name** = enter `pressing_step`.

3. Activate the **SPC** and **LOAD** options.

4. Click the **=** next to **SPC**.

5. Switch **name** to **name(id)**.

This shows the names of the load collectors with their ID numbers in parenthesis.

6. Select the **constraints** load collector.

Note that the field next to the **=** now has a value of 2, which is the ID of the constraints load collector.

7. Click the **=** next to **LOAD** and select the **pressing_load** load collector.

8. Set the **type:** to **linear static**.

9. **Create** the load step **pressing_step**.

In the **status** bar appears the message "The load step has been created." Nothing new is displayed in the graphics area.

10. Click **return** to exit to the main menu.

Step 8: Display the load step on and off (the load collectors defined in the load step).

1. On the **model browser**, turn the display of the load step **pressing_step** off.
2. Notice the load collector's **constraints** and **pressing_load** are no longer displayed.
3. Turn the display of the load step **pressing_step** back on.
4. Click **return** to exit to the main menu.

Section 2: Formatting Models for Analysis

The purpose for using a finite element (FE) pre-processor is to create a model that can be run by a solver. HyperMesh interfaces with many FE solvers and all of them have unique input file formats. HyperMesh has a unique template(s) for each solver it supports. A template contains solver specific formatting instructions, which HyperMesh uses to create an input file for that solver.

This exercise gives you an understanding of the basic concepts for creating a solver input file by using a template. More specifically, you will learn how to review entities in HyperMesh to see how they will appear in the solver input file, define materials and properties and select solver element types for HyperMesh element configurations.

In this section, you will learn how to:

- Create a solver input file by using a template
- Review entities in HyperMesh to see how they will appear in the solver input file
- Define materials and properties
- Select solver element types for HyperMesh element configurations

Interacting with Solvers

- HyperMesh interacts with many solvers
 - Each solver has its own unique formats, terminology, etc.
 - Example: compare nodes and elements in Abaqus and OptiStruct / Nastran
 - 3 nodes
 - 2 quad elements
 - Format / structure is obviously different



Radios (Linear)						Abaqus					
GRID 1 0.0 1.0 0.0 GRID 2 0.0 0.0 0.0 GRID 3 1.0 0.0 0.0 CQUAD4 1 1 1 2 3 4 CQUAD4 2 1 3 4 5 6						*NODE 1, 0.0 , 1.0 , 0.0 2, 0.0 , 0.0 , 0.0 3, 1.0 , 0.0 , 0.0 *ELEMENT,TYPE=S4,ELSET=part_1 1, 1, 2, 3, 4 2, 3, 4, 5, 6					

- HyperMesh can interact with different solvers by using “*templates*”
 - The selected template tells HyperMesh what solver the model is for
 - The template also tells HyperMesh how entities are formatted for that solver
 - Each entity may have several available formats for that solver
 - Each format has fields that make up its definition
 - These fields may need to have values entered by the user
 - Example: a component for Radioss (Linear) can be a PSHELL or PSOLID format
 - PSHELL: holds shell elements, ID = 1, material = 1, thickness 5.0

PSHELL	1	15.0	1	1	0.0
--------	---	------	---	---	-----
 - PSOLID: holds solid elements, ID = 2, material = 1

PSOLID	2	1	0
--------	---	---	---

Tools

- **Collectors** pull-down and **collectors** panels 
 - **Create**
 - Assign a card image to the collector being created
 - Edit the card image fields if desired
 - Assign a material to the collector being created
 - **Update**
 - Assign a material to an existing collector
 - Assign and/or edit a card image of an existing collector
- **Elem types > load types** panels
 - Set a “current element / load type” for an element / load configuration
 - Any new elements / loads of that configuration created will have that element / load type
 - Change the element / load type of existing elements / loads
- **Collectors > Card Edit** or toolbar > 
 - View / edit the card image of any entity in the model
 - Includes entities that are not collectors (nodes, elements, loads, etc.)
- **Model Browser**
 - Right click a collector and select **edit card**
 - View / edit the card image of the selected collector
- **Preferences > graphics**
 - **template labels type**
 - Activate the graphic displayed names of entities in solver (template) terminology rather than HyperMesh (solver neutral) terminology
 - Helps keep track of what is in the model
- **Summary** panel
 - Displays a text window with various types of information about the model
 - Helps to review the model and make sure all information has been entered properly

Summary Type	Included Information
Components	Component Name, ID, Material Name, Thickness, Mass, #Elements
Center of Gravity	Component Name, ID, Mass, X, Y, Z
Elements	Type of Elements, Element Configurations
Error Checks	Load Collector, Load Steps, Components
Loads	Load Collector, ID, FX, FY, FZ, Magnitude
Moment of Inertia	Moment of Inertia

Process

1. Create Materials and enter values into them

2. Use the **model browser** or material collector icon.



Create Properties and enter values into them. Material collectors can be associated with properties during creation.

- Use the **model browser** or property collector icon.



3. Create component collectors and organize entities into them. Property collectors can be associated with components during creation or properties can be assigned directly to elements.

4. Use the **model browser** or component collector icon.



5. Mesh and load the model

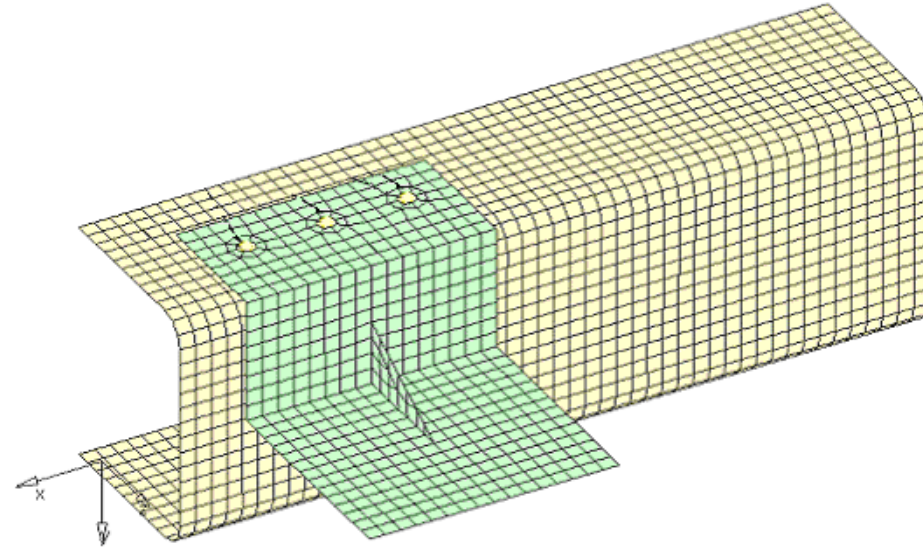
6. Create solver specific cards and controls.

While it is not required that events follow these steps in order, if you follow these steps then there is no need to go back to update or assign information to collectors as the required information will already be in place.

- The goal in formatting for analysis is:
 - All entities have the proper formats (card image / type)
 - Card images of all entities have necessary information entered

Exercise: RADIOSS Linear Statics Setup for a Shell Assembly

This exercise uses the file `channel_brkt_assem_Analysis.hm`. It contains the bracket and channel assembly in the following image.




Step 1: Retrieve and view the file, `channel_brkt_assem_Analysis.hm`.

Step 2: Load the RADIOSS (Bulk Data) template by loading RADIOSS (Bulk Data) user profile.

1. From the **Preferences** menu, click **User Profiles...**
2. Select the **RADIOSS (Bulk Data)** user profile and click **OK** to close the dialog.

Step 3: Review a bracket element to identify what type of RADIOSS element it is and to see how it will be formatted in the RADIOSS input file.

1. Access the **card editor** panel by going to the **collectors** toolbar and selecting **card edit** ().
2. Switch the **entity** selector to **elems**.
3. In the graphics area, select an element of the (blue) bracket.
4. Click **edit**.

The card image for the element appears above the panel menu area. It indicates the element is a RADIOSS CQUAD4 or CTRIA3, depending on whether you selected a quad or tria element. EID is the element's ID. PID is the ID of the element's property. G(X) is the grid (node) ID that makes up the element.

Options specific to the CQUAD4 or CTRIA3 appear in the menu panel area.

5. Return to the **card** panel.

Step 4: Review and edit the existing steel material's card image by accessing the card editor from the model browser.


This material is defined for the channel.

1. In the model browser, open the materials branch of the browser list to show the material steel.
2. Right-click **steel** and select **Card Edit...**
The card image for the material appears. It indicates the material is of RADIOSS Bulk Data type MAT1.
3. Under Poisson's Ratio **[NU]**, change the value from 0 . 3 to 0 . 28.
4. **Return** to the main menu to accept the change.

Step 5: Define a material collector named aluminum for the bracket.


1. Right-click in the white space of the browser list.
2. Select **Create >> Material**.
The **Create Material** popup dialog appears.
3. For **Name:** enter **aluminum**.
4. Set **Card image:** to **MAT1**.
5. Click **Create/Edit** to create the material and edit it.
The card image for the new material appears.
6. Click **[E]** and enter 7 . 0e4 in the field that appears.
This is Young's Modulus.
7. Click **[NU]** and enter 0 . 33 in the field that appears.
This is Poisson's Ratio.
8. Click **return** to exit the panel.

Step 6: Define a Property collector (PSHELL card image) that will be assigned to the Channel component collector.


1. On the **collectors** toolbar, go to the **Property Collector** panel ().
2. Go to the **create** sub-panel.
3. For **name=** enter **channel**.
4. Click the **type=** button and select **2D**.
5. Click the **card image=** button and pick **PSHELL**.
6. Click the **material=** button and select the **Steel** material collector.
7. Click the green **create/edit** button (This will open the card editor panel).

8. Click **[T]** and enter 3.0 into the field that appears. (This will assign a 3.0 unit thickness).
9. **Return** to the main menu.

Step 7: Assign the *channel* property to the channel component.

1. On the **collectors** toolbar, go to the **Components panel** ().
2. Go to the **assign** sub-panel.
3. Select **comps >> channel**.
4. Click **select** to complete the selection.
5. For **property=** select **channel**.
6. Click **assign**.
7. Click **return** to exit the panel.

Step 8: Update the bracket property to have a PSHELL card image, a thickness of 2.0, and the aluminum material.

1. Access the **properties** panel in one of the following ways:
 - From the **Pull-down** menu, select **Properties**, then **Create**.
 - From the **Collectors** toolbar, select **properties** (.
2. Go to the **update** sub-panel.
3. Select **props >> bracket**. (Click **props** to select from the component list.)
4. Click **select** to complete the selection.
5. For **card image =** select **PSHELL**.
6. For **material =** select **aluminum**. (Click the text field to select from the material list.)
7. Click **update/edit** to load and edit the card image and assign the material.
8. Notice the material ID MID is 2, which is the ID of the aluminum material you created earlier and assigned to the bracket component.
9. For the thickness **[T]** enter 2.0.
10. **Return** to the **properties** panel.
11. **Return** to the main menu.

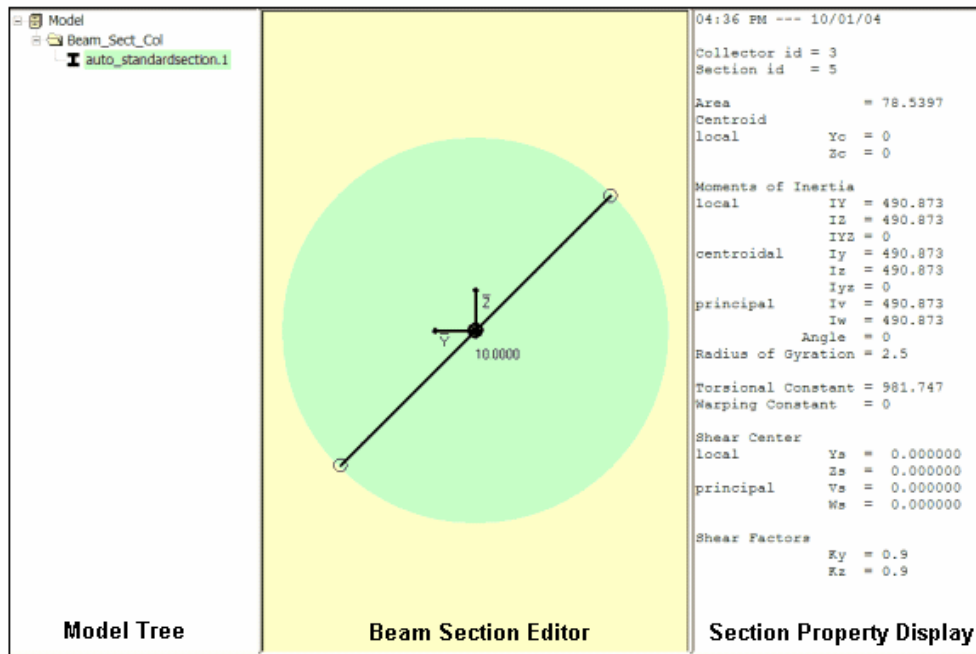
Step 9: Calculate the section properties for the bar elements (RADIOSS CBEAM) by using HyperBeam.

1. On the **1D** page, enter the **HyperBeam** panel.
2. Go to the **standard section** sub-panel.
3. Switch the **standard section type:** to **solid circle**.

- Click **create** to invoke the HyperBeam module.

The HyperBeam module appears and the HyperMesh session is not visible. (HyperMesh is visible again upon exiting HyperBeam.)

The solid, red circle represents the cross section. Under the local coordinate system you should see the number, 10.0000, which is the circle's diameter.



HyperBeam module with the standard solid circle section

- Click on the number **10.0000** to activate a number entry field, enter 6, and then press ENTER.

In the section property display area, the values are automatically updated to reflect the circle's new diameter.

- In the model tree area, click on the section's name, **auto_standardsection_1**.

After a brief moment, the name becomes an editable text entry field.

- Rename the section to **6mm_Beam_Sect**.
- From the **File** pull-down menu, select **Exit**.
- Answer **Yes** to the question "save data?".
- Return** to the main menu.

Step 10: Create a property collector named bars_prop for the bar elements (RADIOSS Bulk Data).

- Create a new property collector by:
 - Using the **properties: create** sub-panel
 - Right-clicking in the model browser

2. Enter the name `bars_prop`.
3. Specify the card image.
 - For the **properties** panel, set **type=** to **1D** and switch **card image=** to **PBEAM**.
 - For the **Create Properties** dialog, set **Card Image:** to **PBEAM**.
4. Set the **material** to **steel**.
5. Click **create/edit** to create the property and edit it.
6. At the top of the card image, select **beamsec >> 6mm_Beam_Sect**.

The parameter fields in the PBEAM card are automatically populated by the data in the beam section 6mm_Beam_Sect.

7. **Return** to the main menu.

Step 11: Update the CBEAM element to use the PBEAM Property.

1. Access the **Properties: assign** sub-panel.
2. Select **elems>>by collector** and pick the **bolts** component collector.
3. Click the green **select** button.
4. Set **type=** to **1D**.
5. Click the **property=** button and select the **bars_prop** property collector.
6. Click the green **assign** button.
7. **Return** to exit the panel.

Step 12: Define a H3D file to be output from RADIOSS by using the **control cards** panel.

1. Access the **control cards** panel in one of the following ways:
 - From the **Pull-down** menu, select **Setup**, then **Create**, then **Control Cards**
 - On the main menu, select the **Analysis** page, then select **control cards**
2. Select the control card **FORMAT**.

You may need to click **next** to get to the second page of cards.

Notice in the card image the one **FORMAT** line is set to **H3D**. This specifies RADIOSS to output results to a Hyper3D (H3D) file, which can be viewed in HyperView Player. Also, an HTML report file will be output and the H3D file will be embedded in it.

3. For **number_of_formats =** specify 2.
A second **FORMAT** line appears in the card image.
4. Click **H3D** in the second line of the card image and select **HM**.

This specifies RADIOSS to output the results to a HyperMesh binary results file, allowing the results to be post-processed within HyperMesh.

- Click **return** to exit to the **control cards** panel.

Notice the FORMAT button is green. This indicates the card will be exported to the RADIOSS input file.

- Click **return** to exit to the main menu.

Step 13: Export the model to a RADIOSS Bulk Data input file.

- On the **File** menu, click **Export**.
- In the **Export** tab, for **Export Type**: specify **FE Model**.
- In the **File name**: field, type `channel_brkt_assem_loading.fem`.

Note that the extension for an RADIOSS Bulk Data input file is `.fem`.

- Click **Save** to export the model as an RADIOSS `.fem` input file.

This exports the model as an input file for the solver specified by the current user profile.

Step 14: Review the contents of the file `channel_brkt_assem_loading.fem`.

- Using any text editor (Notepad, Wordpad, Vi, etc.), open the file `channel_brkt_assem_loading.fem`.
- Near the top of the file, notice as shown in the following image:
 - The line `FORMAT HM` which you specified in HyperMesh
 - The load step (RADIOSS Bulk Data SUBCASE) named `pressing_step` which you defined in HyperMesh
 - Under the load step, the load collector ids (RADIOSS load and constraint set identification numbers)

```

FORMAT H3D
FORMAT HM
$$$$-----$
$$$                      Case Control Cards                      $
$$$$-----$
$
$HMNAME LOADSTEP          1"pressing_step"          1
$
SUBCASE          1
  SPC =          2
  LOAD =         1

```

- Search for "FORCE".

Notice the load set identification number for each force (RADIOSS FORCE). It is either 1 or 2 as shown in the following image. These numbers correspond to the numbers under the load steps in the file.

```

$  FORCE Data
$
FORCE          1    2587    01.0    0.0    5.0    0.0
FORCE          1    2571    01.0    0.0    5.0    0.0

```

- Search for "SPC" (HyperMesh constraint).

Notice the constraint set identification number for each constraint (RADIOSS SPC). It is 2 as shown in the following image, which lists a few of the constraints. This number corresponds to the number under the load steps in the file.

```

$$ SPC Data
$$
SPC          2      59 1234560.0
SPC          2     698 1234560.0
SPC          2     699 1234560.0
SPC          2     700 1234560.0
SPC          2     701 1234560.0
SPC          2     702 1234560.0

```

5. Search for the load collector name "pressing_load".

Notice the load collectors pressing_load and constraints. Also, notice their collector ID and color ID. When the model is imported into HyperMesh, the loads are organized into these load collectors and have these IDs and colors.

```

$$$$-----$
$$$ HyperMesh Commands for loadcollectors name and color information $
$$$$-----$
$HMNAME LOADCOL          1"pressing_load"
$HWCOLOR LOADCOL         1      5
$$$
$HMNAME LOADCOL          2"constraints"
$HWCOLOR LOADCOL         2     49

```

6. Close the file channel_brkt_assem_loading.fem.

Step 15 (Optional): Save your work.

With the exercise completed, you can save the model as a HyperMesh file, if desired.

Chapter 7

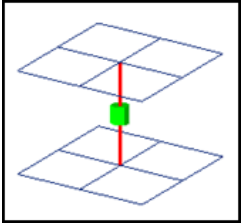
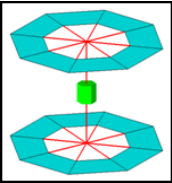
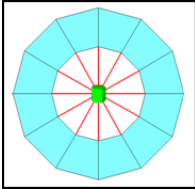
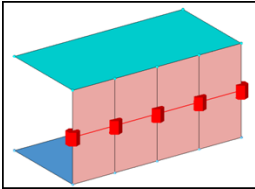
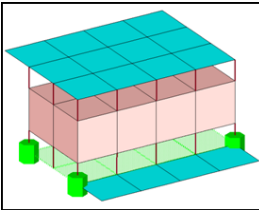
Connectors

Analyzing an assembly of parts is a common need. The parts in the assemblies need to be held together, whether by bolts, welds, rivets, or adhesives. There are many different ways to model these connection types in FEA, which are specific to the type of analysis being done. The connectors functionality in HyperMesh allows them to be created and managed easily, especially when large numbers are involved, for instance with welding. This section examines the different types of connections and how to create and edit them within HyperMesh.

- In this chapter, you will learn how to:
- Weld the two front trusses to each other by creating connectors at pre-defined weld points.
- Weld the two front trusses to the reinforcement plate by creating connectors between shell elements.
- Weld the right rails to each other and to the front trusses by creating connectors from a master connectors file.
- Apply an adhesive connection to the left rails.
- Apply a bolted connection to the two rear trusses.

Connectors: What are they

Connectors are a geometric representation of connections between entities. The advantage of connectors is the ability to create multiple connections at a single time. Hundreds or even thousands of connections that would normally have to be created manually, one at a time can be mass created, even before the part is meshed. They can be used to create numerous types of connection elements such as;

- **Spot Welds** 
- **Bolts** 
- **Trim Masses** 
- **Seam Welds** 
- **Area Connections (Adhesives)** 

Connectors: Terminology

- **Link Entities** - The entities that are being connected
 - User can explicitly define link entities or specify a search tolerance
 - Can be components, elements, surfaces, nodes, or tags
 - Typically components are linked
- **Connector Location** - Where the entities are linked
 - Nodes – created at the node location
 - Points – created at the point location
 - Lines – created on the line

- The line may be split into multiple projection locations as specified by the offset, spacing, and density values
 - Elements – created at the element location (adhesives only)
 - Surface – created at the surface location (adhesives only)
- Connector Realization** – The creation of the finite element representation of that connector
 - Rigids, springs, etc., or custom configurations such as ACMs, CWELDS, etc.
- Connector State** – Whether an FE representation of a connector has been created
 - Unrealized* - The initial status of the connector entity upon creation
 - Realized* - The status only if creation of the FE weld representation at the connector was successful
 - Failed* – The status if creating the FE weld representation at the connector was not successful
- # of Layers** – number of FE weld layers to attempt to generate for the connector
 - 2T, 3T, etc.
- Connect When:** – Specifies when the link entity information is added to the connector
 - Now* - Allows you to add link entity information now. For this option, you must select the *connect what* entities and *num layers* to successfully create a connector.
 - At FE Realize* - The link entities to the connector are determined while realizing the connector. The link entities are determined by the projections and proximity from the connector location.
 - Re-Connect Rule* – Defines method for connector re-attachment during part swapping/replacement
 - None* - If a link entity is deleted, the link entity is removed from the connector
 - By ID* - If a link entity is deleted, the connector retains the ID of the link entity, and will to a new entity with that ID upon realization
 - By Name* – Same as the *by id* rule except that the entity name is retained

Connectors: Tools

- Connectors Module**

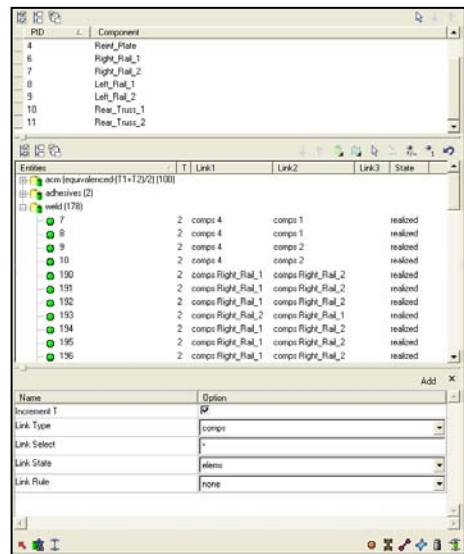
- Spot**
 - Bolt**
 - Seam**
 - Area**


create, edit, and realize
that style of connectors

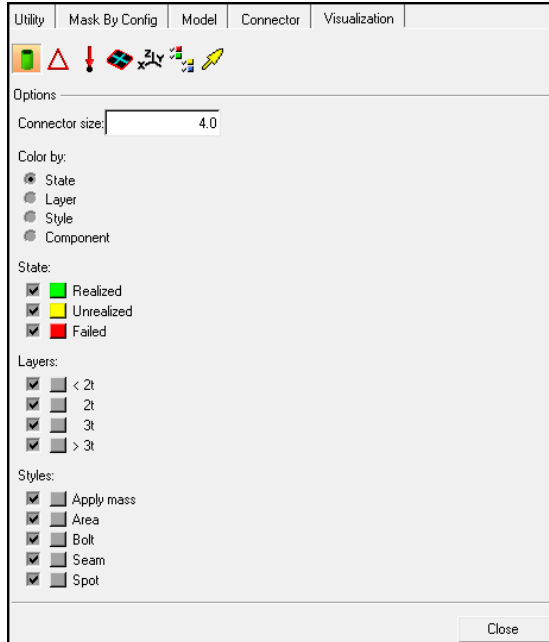
spot	add links	compare
bolt	unrealize	quality
seam		
area		
apply mass		
fe absorb		

- Apply Mass** – adds a mass value to entities
 - Used to represent mass of parts that are not present in the model
 - FE Absorb** – Create new connectors from existing elements of recognizable FE representations of welds, bolts, adhesives, etc.
 - Add Links** – Add link entities to existing connectors
 - Unrealize** – Delete FE representations of welds / bolts / adhesives associated with existing connectors
 - Compare** – Checks the MCF against displayed model file

- **Quality** – Check for duplicate connectors, combines connectors, and checks the quality of realized elements
- **Connector Browser** – Hierarchical browser that provides information and the ability to edit connectors.
 - Shows:
 - Type of Connector
 - Link Information
 - State of connector
 - Editable
 - Edit link entities, export mwf files, etc
 - Found in Tab Browser Area

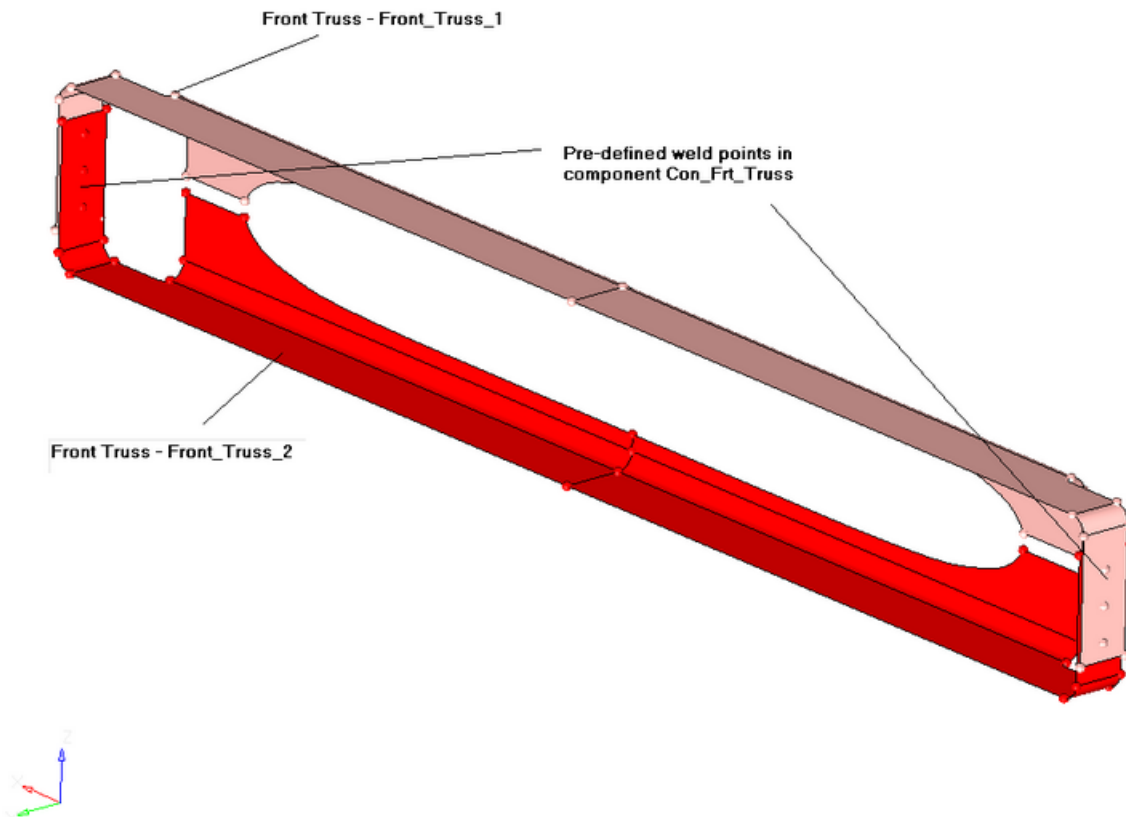


- **Visualization** –  Controls how connectors are displayed:
 - Color connectors by state, layers, or component
 - Visibility by state or layers (can turn the display on or off)
 - Control size of connector display



Exercise 1: Connectors Introduction using Spot Connectors


This exercise uses the model file, `frame_assembly.hm`. Weld the two front trusses by creating connectors between geometry surfaces at pre-defined weld points.





Step 1: Retrieve and view the model file.

1. Open the model file, `frame_assembly.hm`.
2. Take a few moments to observe the model using various visual options available in HyperMesh (rotation, zooming, etc.).
3. From the **Preferences** pull-down menu, select **User Profiles**.
4. Select **RADIOSS**, then **Bulk Data**.
5. Click **OK** to load the user profile.

Step 2: Display only the assembly, `assem_1` for elements and geometry.

1. Click the **Model** tab in the **tab** area. Or, go to the **View** menu and click **Model Browser**.
2. Select the **Model View** icon, , in the **Model Browser** and expand the **Assembly Hierarchy** tree.

3. Right-click the **entity selection**, , and select the **Elements + Geometry** icon, .

This turns on/off both elements and geometry when performing right-click operations in the **model browser**.

4. Right-click **assem_1** and select **Isolate**.

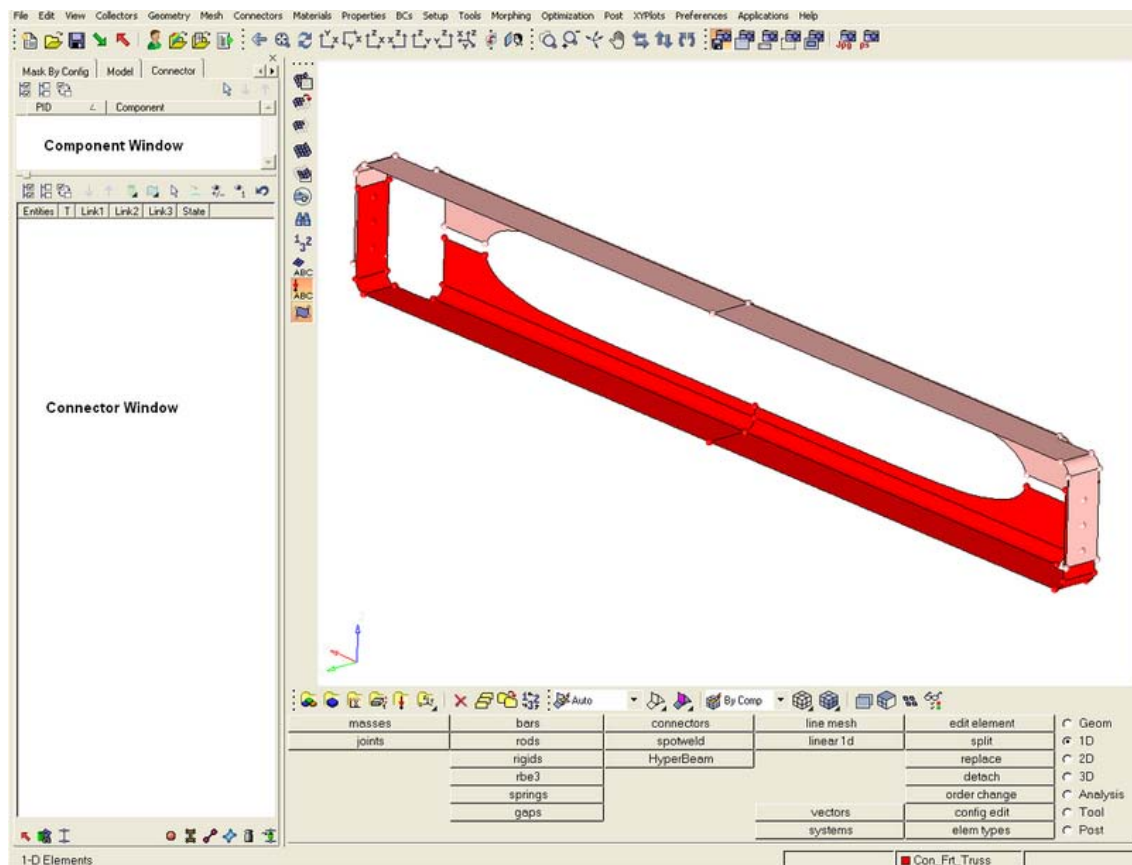
This will only display the components that are in the **assem_1** assembly.

Step 3: Load the Connector Browser.

1. Go to the **View** menu and click **Connector Browser**.
2. Review the layout of the **Connector Browser**.

The **Connector Browser** allows users to view and manage the connectors in their model. The top portion of the browser is called the **component window** and it contains a listing of the components being connected with the connectors. The lower portion is called the **connector window** and it contains a list of the connectors in the model. The connectors will be grouped based on the type of connection.


Currently there are no components or connectors listed because there are no connectors in the model.

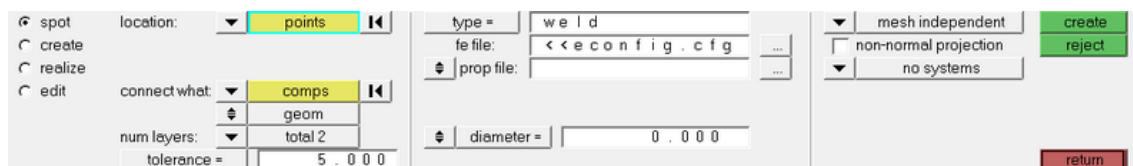


Step 4: Create welds between the geometry for the two front trusses at the pre-defined weld points.

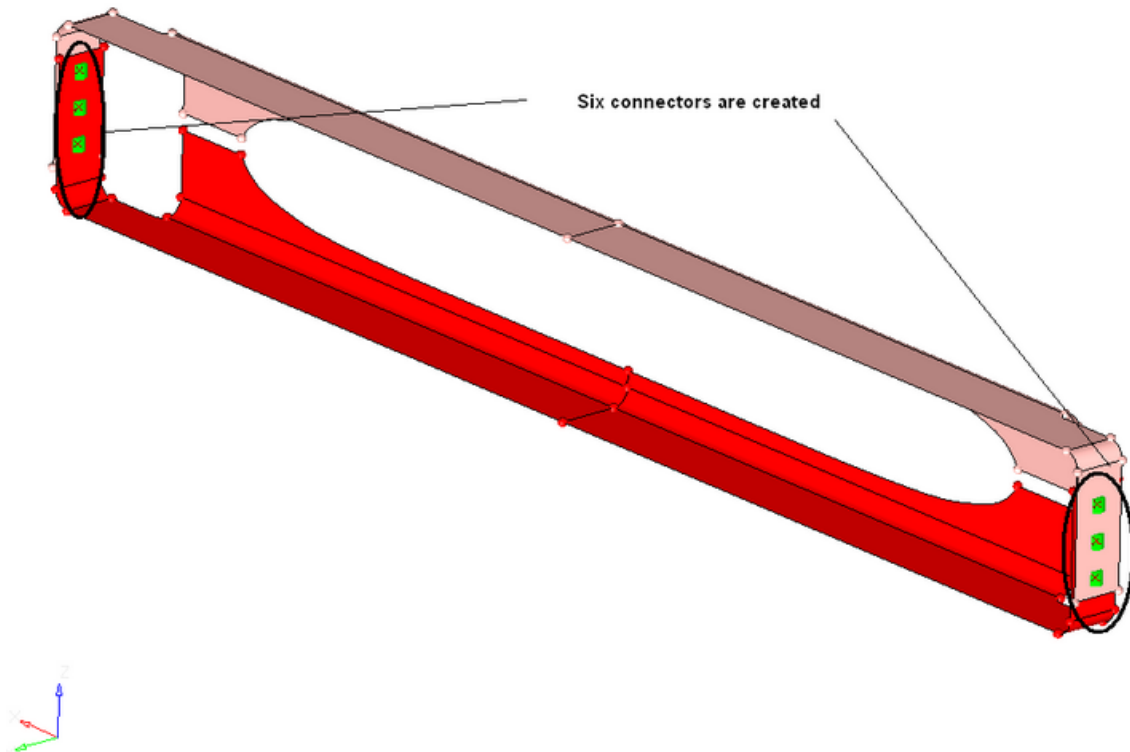
There are two methods for creating connectors: automatic and manual. The automatic approach creates and realizes connectors automatically. The manual approach allows you to create the connector manually, and then realize the connector manually. Realization is the process in which the connector creates the weld entity.

To create connectors automatically, access the **spot**, **bolt**, **seam**, and **area** panels within the **connectors** module. To create connectors manually, go to the create and realize sub-panels.

1. Access the **spot** panel in one of the following ways:
 - Right-click in the **Connector Browser** in the **connector window** and select **Create >> spot**
 - From the main menu, select the **1D** page, then select the **connectors** module, and then select **spot**
 - At the bottom of the **Connector Browser** click **spot** ()
2. Access the **spot** sub-panel.
3. Verify that the current component is **Con_Frt_Truss**.
4. Switch the **location: entity selector** to **points**.
5. Select the six pre-defined weld points by selecting **points >> by collector** and selecting the component **Con_Frt_Truss**.
6. Click **select**.
7. For **connect what:**, click **comps** and select the components **Front_Truss_1** and **Front_Truss_2**.
8. For **connect what:**, toggle **elems** to **geom**.
9. For **tolerance** = specify 5. The connector will connect any selected entities within this distance of itself.
10. For **type**=, select **weld**.
11. Click **create**.
12. Click **return**.



The six connectors are automatically created and realized (notice the message in the **status** bar). The green connectors indicate that the creation of the weld entity was successful. The connectors are organized as geometry (not elements) in the current component collector, **Con_Frt_Truss**.



There are three states of connectors: realized (green ■), unrealized (yellow ■) and failed (red ■). The color of the connectors change from yellow to green (if created manually), indicating they are realized into weld elements. As mentioned before, if created automatically they will be green immediately as there is no interim unrealized (yellow) state.

Fixed points were added to the surfaces at the ends of the weld elements to guarantee connectivity between the weld elements and the shell mesh that will be created on the surfaces.



Weld element with fixed points created on the surfaces

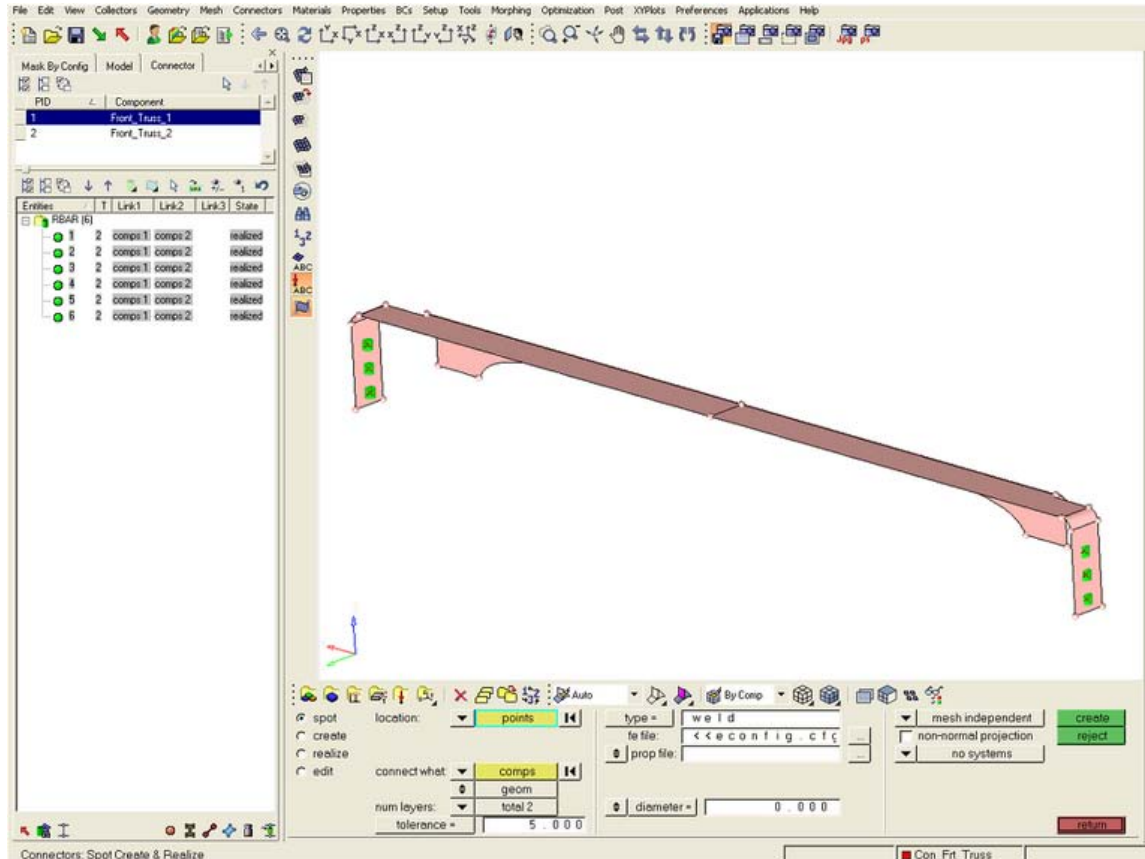
Step 5: Review the Connector Browser.

1. In the **connector window** in the **Connector Browser**, click the **+** next to **RBAR**.

This contains a listing of the six connectors just created. They are all grouped under **RBAR** because that is the type of connector created. Notice the ID of the connector under **Entities**, the **Links** of the connector, and the **State** of the connector. You may need to increase the size of the **tab** area to the right to see the **State** column.

2. In the **collector window** in the **Connector Browser**, right-click **Front_Truss_1** and select **find**.

There are five different options available. Select **Find**. This isolates this collector in the **graphics area**. It also highlights the six connectors in the **connectors window** to indicate that these connectors have **Front_Truss_1** as a link.



3. Right-click **Front_Truss_1** again, and this time select **Find Attached**.

This will find the components that are attached to **Front_Truss_1** through connectors. Notice that **Front_Truss_1** and **Front_Truss_2** are now both highlighted in the component window to indicate that they have been displayed in the **graphics area**.

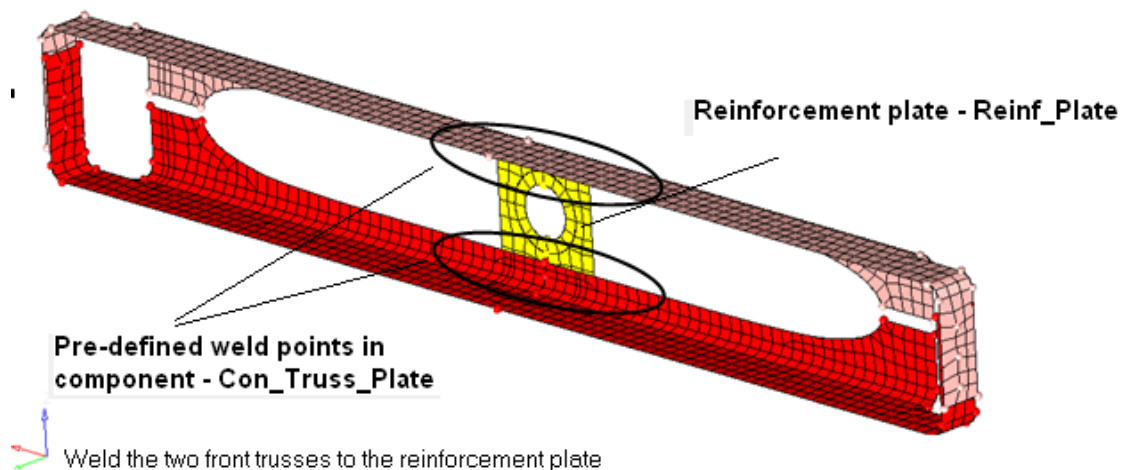
Step 6: Create a shell mesh on the two front truss components.


1. Press F12 to go to the **automesh** panel.
2. Access the **size and bias** sub-panel.
3. Select the mesh mode **automatic**. (It currently may be **interactive**.)
4. Select **surfs >> displayed**.
5. For **elem size** =, specify 10.
6. For **mesh type**, select **mixed**.
7. Verify that the toggle is set to **elems to surf comp**. (It currently may be **elems to current comp**.)

8. Click **mesh** to mesh the surfaces.
9. Zoom into the area with a connector and see how the fixed point created from the weld has ensured the mesh seeding passes through the weld.
10. Click **return** to return to the main menu.

Step 7: Display only the assembly **assem_2** for elements and geometry.

Weld the two front trusses to the reinforcement plate by creating connectors between shell elements at pre-defined weld points.



1. Click the **Model** tab in the **tab** area. Or, go to the **View** menu and click **Model Browser**.
2. At the top of the browser window, verify that **Elements + Geometry** () is the current selection type.
3. Right-click **assem_2** and select **Isolate**.

Step 8: Create connectors between the shell mesh for the front trusses and the reinforcement plate at pre-defined points.

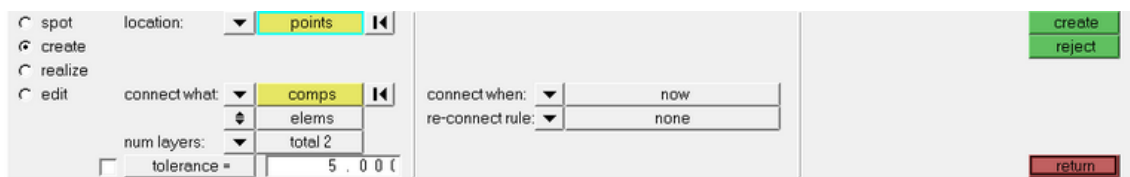
Perform the following steps to create and realize the connectors manually.

1. On the **status** bar, click the **set current component** and set the current collector to **Con_Truss_Plate**.
2. Go to the **Connector** tab in the **tab** area.
3. Right-click in the **connector window** and select **Create >>Spot**.
4. Access the **create** sub-panel.
5. For **location:**, select **points**.
6. Select **points >> by collector** and select the component **Con_Truss_Plate**.
7. Click **select**.
8. For connect **what:**, select the following components:

- **Front_Truss_1**
- **Front_Truss_2**
- **Reinf_Plate**

- Click **select**.
- Also for **connect what:**, toggle **geom to elems**.
- Verify that **num layers** is set to **total 2**.
- Also verify that **connect when:** is set to **now**.
- Click **create** to create connectors at the selected weld points.

Notice that the status bar says "8 spot connectors created with comps and links." The connectors are organized into the current component collector, **Con_Truss_Plate**.



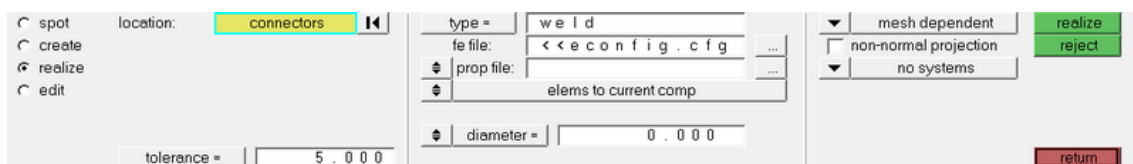
- Notice in the **Connector Browser's connector window** that these eight connectors are currently grouped as **undefined** and that the color of the connectors is yellow to indicate they are unrealized.
- Click the **+** next to **undefined** and notice the status of the eight connectors is unrealized. In the next step, you will realize the connector and assign a connector type.

Step 9: Realize the connectors in the component **Con_Truss_Plate** into weld elements.

- Go to the **realize** sub-panel.
- Select **connectors >> displayed**.
- For **type=**, select **weld**.
- For **tolerance =**, enter 5.
- Verify the selection is set to **mesh dependent** from **mesh independent**.

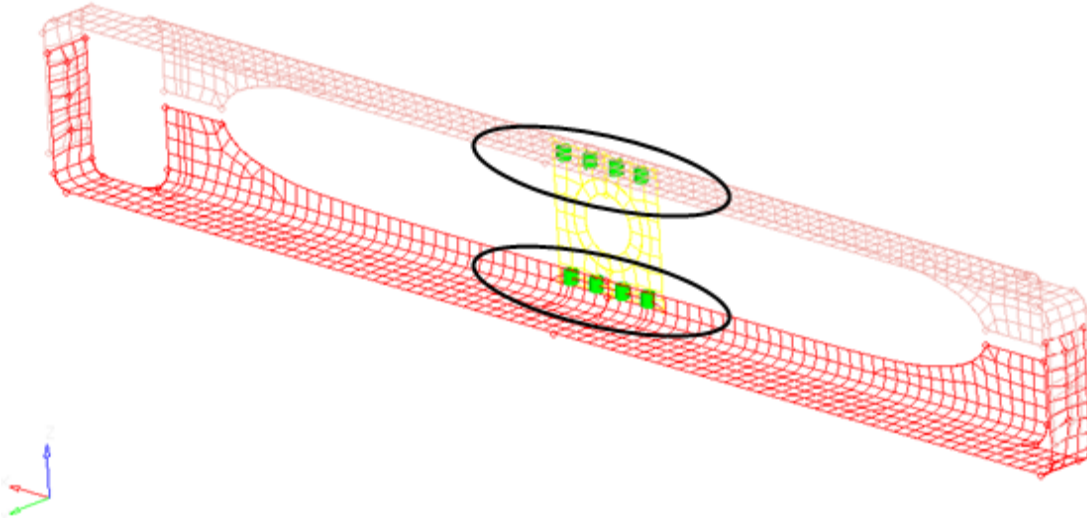
When the option **mesh dependent** is active, if the realized finite element of the connector is coincident to a node of the shell mesh it is being connected to, the nodes are equivalenced. If there are no suitable nodes present, this option will partition the mesh accordingly to ensure the mesh seeding passes through the weld point.

- Click **realize** to realize the selected connectors into weld elements.

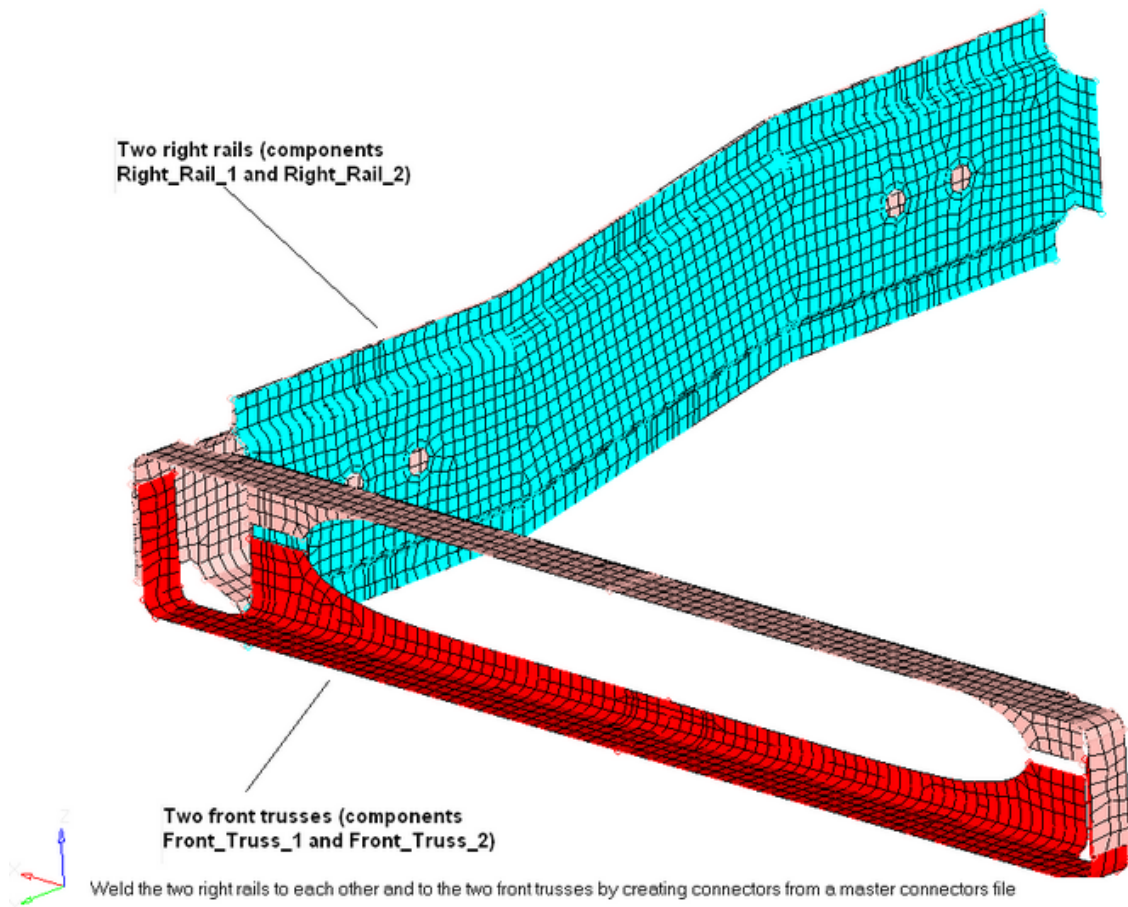


- Review the **Connector Browser**. Notice the following:
 - **Reinf_Plate** is now a component listed in the **component window**.


- The eight connectors that were just realized are now grouped under **RBAR** in the **connector window** instead of **undefined** as we just assigned the connector **type**.
8. Zoom into the eight connectors that were just realized. Notice how the mesh was partitioned to ensure the mesh seeding passed through the weld points.
 9. Click **return** to return to the main menu.




The two front trusses welded to the reinforcement plate with weld elements at the connectors




Step 10: Display only the assembly *assem_3* for elements and geometry.

10. 1. Click the **Model** tab in the tab area. Or, go to the **View** menu and click **Model Browser**.
11. 2. At the top of the browser window, verify that **Elements + Geometry** () is the current selection type.
12. 3. Right-click **assem_3** and select **Isolate**.

Step 11: Create connectors to connect the right rails to each other and to the front trusses by importing a master connectors file.

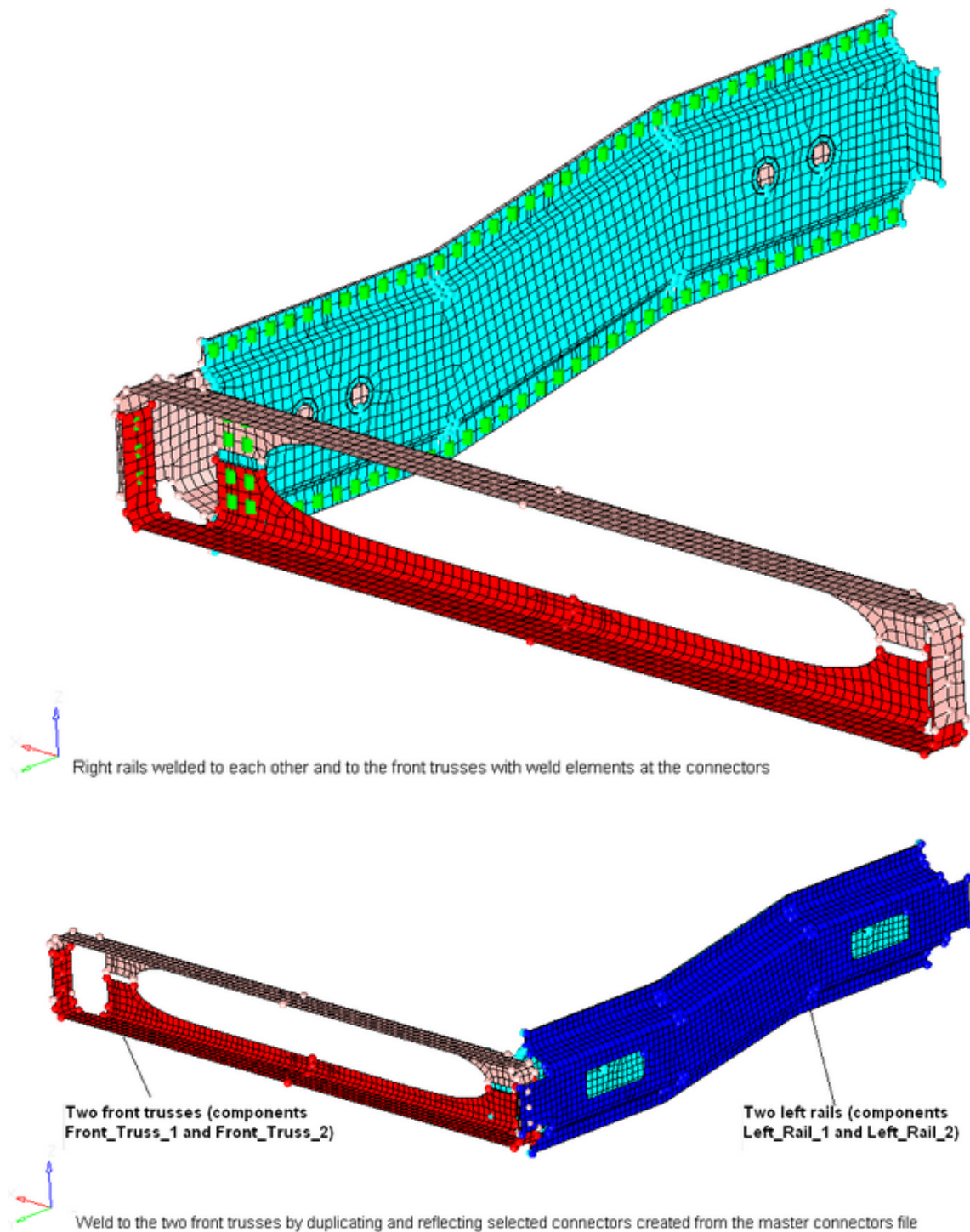
1. Access the **Import** Tab in one of the following ways:
 - From the **Pull-down** menu, select **File**, then select **Import**.
 - In the **Standard** Toolbar select **Import** (.
2. For **Import type**, select **Connectors**.
3. For **File type**, select **Connectors**.

4. Click **Open** () and import the file, `rails_frt_truss.mwf`.
5. Click **Apply**.

It will take a few seconds for the connectors to be imported. The connectors are automatically created and are organized into the new component, **CE_Locations**.


Step 12: Realize the connectors in the component **CE_Locations** into weld elements.

1. In the **status** bar, click **set current component** and set the current component collector to **CE_Locations**.
2. Click the **Connector** Tab. Right-click in the connector window and select **Create >> spot**.
3. Access the **realize** sub-panel.
4. Select **connectors >> displayed**.
5. Verify **type=** is set to **weld**.
6. Verify **tolerance =** is set to 5.
7. Verify **mesh dependant** is selected.
8. Click **realize** to realize the selected connectors into weld elements.
9. Click **return** to return to the main menu.




Step 13: Display only the assembly *assem_4* for elements and geometry.

1. Click the **Model** tab in the **tab** area. Or, go to the **View** menu and click **Model Browser**.

2. At the top of the browser window, verify that **Elements + Geometry** () is the current selection type.
3. Right-click **assem_4** and select **Isolate**.

Step 14: Create a new component collector to hold new connectors.

1. Enter the **component collectors** panel by one of the following ways:
 - From the **collectors** toolbar, click **Components** ()
 - From the **Pull-down** menu, select **Collectors**, then select **Create** and then select **Components**
2. Verify you are in the **create** sub-panel.
3. For **name** = specify **CE_Locations_Dup**.
4. Select a color for the component.
5. Toggle **property=** to **<property blank>**.
6. Click **create** to create the component.
7. Click **return** to return to the main menu.

Step 15: Duplicate the connectors created from the master connectors file and reflect them.

1. Enter the **reflect** panel by one of the following ways:
 - From the **Pull-down** menu, select **Connectors**, then select **Reflect** and then select **Connectors**
 - From the main menu, select the **Tool** page, then select **reflect**
2. Verify the **entity** selector is set to **connectors**.
3. Using the **model browser**, turn on and display the component, **CE_Locations**.
4. Select **connectors >> by collector** and select **CE_Locations**.
5. Click **connectors >> duplicate >> current comp**.

The displayed connectors are duplicated and the duplicates are organized into the current component, **CE_Locations_Dup**.
6. Set the **plane** selector to **x-axis**.

This is the axis normal to the plane of interest.
7. Click **B** twice to specify a **base** node to reflect about.
8. Click **x=**.

The fields for **x=**, **y=**, and **z=** are activated. By default, their value is 0.000, which is the base point you want to reflect about.
9. Click **return** to return to the **reflect** panel.
10. Click **reflect** to reflect the connectors and notice they are unrealized (yellow).

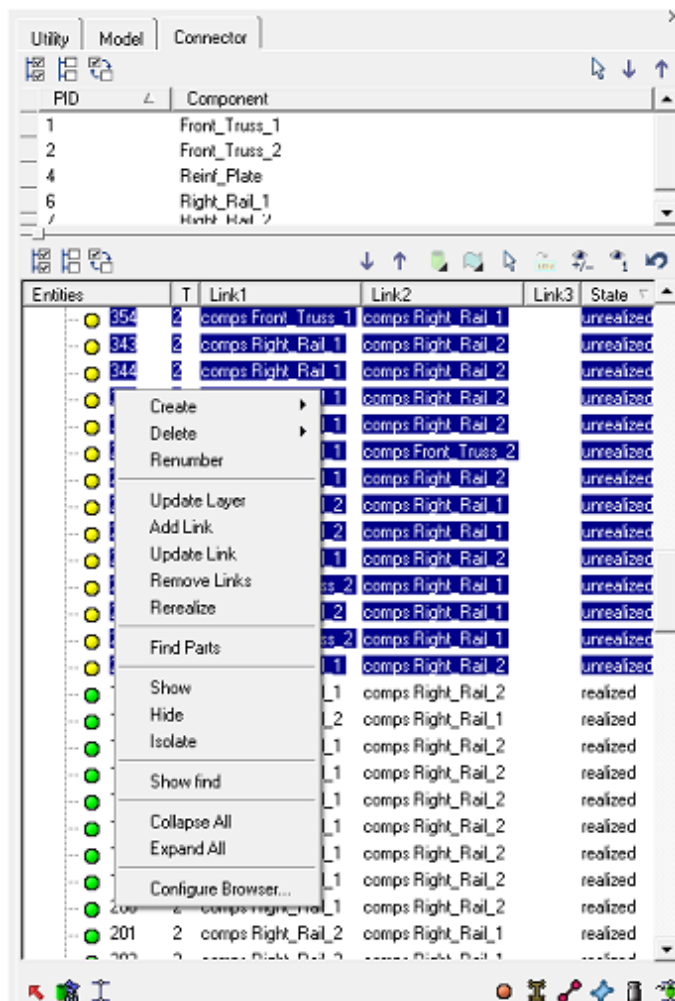
- Click **return** to return to the main menu.

Step 16: Update the connectors for the left rails to link them to the left rail components.

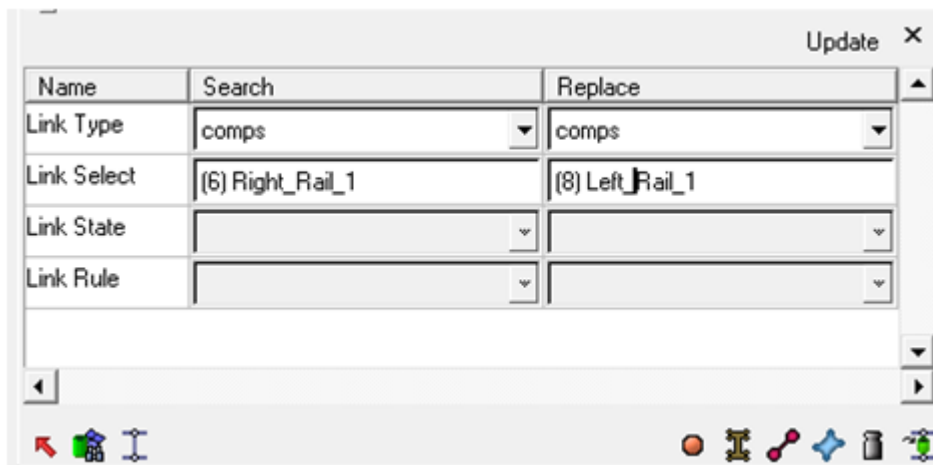
- Enter the **Connector Browser**.
- Expand the folder for **RBAR** in the connector window.
- Click **State** to sort the connectors by their state.

You may need to increase the size of the **Tab Area** to the right to see the **State** column. Notice how all the realized connectors are now at the top of the list.

- Click **State** again so that all the unrealized connectors are at the top of the list.
- Left-click the first connector in the list under the **Entities** column. Scroll down to the last unrealized connector in the list. While holding the SHIFT button, left-click the last unrealized connector in the list. This will select all the unrealized connectors in the list. Verify that this is the case.
- Right-click under **Entities** and select **Update Link**. This brings up the **update window** under the **connector window** in the **Connector Browser**.



7. Review the entries in the **Link1** and **Link2** columns.
8. Notice that the connectors have the links comp **Right_Rail_1** and comp **Right_Rail_2**. This data is from the master connectors file that you imported. These links need to be updated to reflect the two left rails.
9. Set **Link Type** to **comps** under **Search** in the **update window**.
10. Click in **Link Select** under **Search**. This brings up a component selector in the panel area.
11. Click **component** and select the component, **Right_Rail_1**.
12. Click **proceed**. This updates the **Connector Browser** so now the **Link Select** is set to **Right_Rail_1** under **Search**.
13. Set **Link Type** to **comps** under **Replace**.
14. Click in **Link Select** under **Replace**. This brings up a **component** selector in the panel area.
15. Click **component** and select the component, **Left_Rail_1**.
16. Click **proceed**. This updates the **Connector Browser** so now the **Link Select** is set to **Left_Rail_1** under **Replace**.



17. Click **Update** to update the connectors' links.
18. Repeat #9 through #19, except search for the **Right_Rail_2** component and replace it with the **Left_Rail_2** component.
19. Scroll through the list of unrealized connectors to make sure that no connector is linked to the right rail components.
20. Click the **X** next to **Update** to close the **update window**.

Step 17: Realize the connectors in the component, **CE_Locations_Dup**, into weld elements.


1. Go to the **spot: realize** sub-panel.
2. Select **connectors >> by collector** and select the component, **CE_Locations_Dup**.
3. Verify **type=** is set to **weld**.

4. Verify **tolerance** = is set to 5.
5. Click **mesh dependent** option is active.
6. Click **realize** to realize the selected connectors into weld elements.

Step 18: Verify that all connectors are realized and identify the pairs of adjacent connectors.

1. In the **connector window** of the **Connector Browser**, verify that the **RBAR** folder is expanded.
2. Scroll through all the connectors. Note that the **State** column indicates that all of the connectors are realized.
3. Zoom into one of the two areas where the front trusses are connected to the rail components.


Note that at these two areas, there are pairs of adjacent connectors.


4. Click **Visualization Options** () from the **visualization** toolbar.
5. On the **Connectors** tab, select **Layer** for the **color by:** option.
6. While still in the **visualization** dialog, notice under **layers** that **2t** (two thickness) is purple.

The connectors are now colored purple. This means each of these connectors link two components.

Because the pairs of connectors create a series of two weld elements, you can combine each pair into a single connector, which links the three components together.

Step 19: Isolate the pairs of adjacent, 2t connectors identified in the previous step.

1. From the **model browser**, turn off the display for all geometry components.
2. Enter the **find** panel by doing one of the following:
 - From the **Pull-down** menu, select **Edit**, then **Find**
 - From the main menu, select the **Tool** page, then select **find**
3. Select the **between** sub-panel.
4. Switch the **entity type** to **connectors**.
5. Switch the **entity** selector to **comps**.
6. Select the components **Front_Truss_1** and **Front_Truss_2**.
7. Click **select**.
8. Click **find** to find the six connectors between these components.
9. Next to the **comps** selector, click  to reset the selection.
10. Select the components, **Front_Truss_1** and **Right_Rail_1**.

11. Click **find** to find the three connectors between these components.
12. Click .
13. Select the components **Front_Truss_1** and **Left_Rail_1**.
14. Click **find** to find the three connectors between these components.
15. Click **return** to return to the main menu.

Step 20: Unrealize the displayed connectors.

1. Access the **unrealize** panel by doing one of the following:
 - From the **Pull-down** menu, select **Connectors**, then **Unrealize**
 - From the main menu, select the **1D** page, then select **connectors**, then select **unrealize**
2. Select **connectors >> displayed**.

Notice the **status** bar displays "12 connectors added by 'displayed'. Total selected 12."
3. Click **unrealize** to unrealize the connectors.

The weld elements associated to these connectors are deleted.
4. Click **return** to return to the main menu.

Step 21: Combine the pairs of adjacent 2t connectors into 3t connectors.

1. Enter the **connector quality** panel by doing one of the following:

From the **Pull-down** menu, select **Connectors**, then select **Check**, then **Connectors**, then **Quality**


From the main menu, select the **1D** page, then select **connectors**, then **quality**
2. In the **connectors** (unrealized) sub-panel, select **connectors >> displayed**.
3. For **tolerance** = specify 5.
4. Click **preview combine**.

The **status bar** displays the message, "12 connector(s) found that need to be combined."
5. Click **combine** to combine the connectors.


The **status** bar displays the message, "6 connectors deleted." Also notice that the connectors have now turned a dark blue color to indicate that they have 3 layers.
6. Click **return** to return to the main menu.

Step 22: Realize the 3t connectors in the component, **Con_Frt_Truss**, into weld elements.

1. Set the current component to **Con_Frt_Truss**.
2. Enter the **spot: realize** sub-panel.


3. Select **connectors >> displayed**.
4. Verify **element config:** is set to **weld**.
5. Verify **tolerance =** is set to 5.
6. Toggle **mesh dependent** option to be active.
7. Click **realize** to realize the connectors.
8. Click in the **Connector Browser**. Scroll down through the list in the **connector window** to see that there are now three links for the six connectors we just updated.
9. Click **return** to return to the main menu.
10. Click **Visualization Options** () from the **visualization** toolbar.
11. On the **Connectors** tab, select **State** for the **color by:** option.

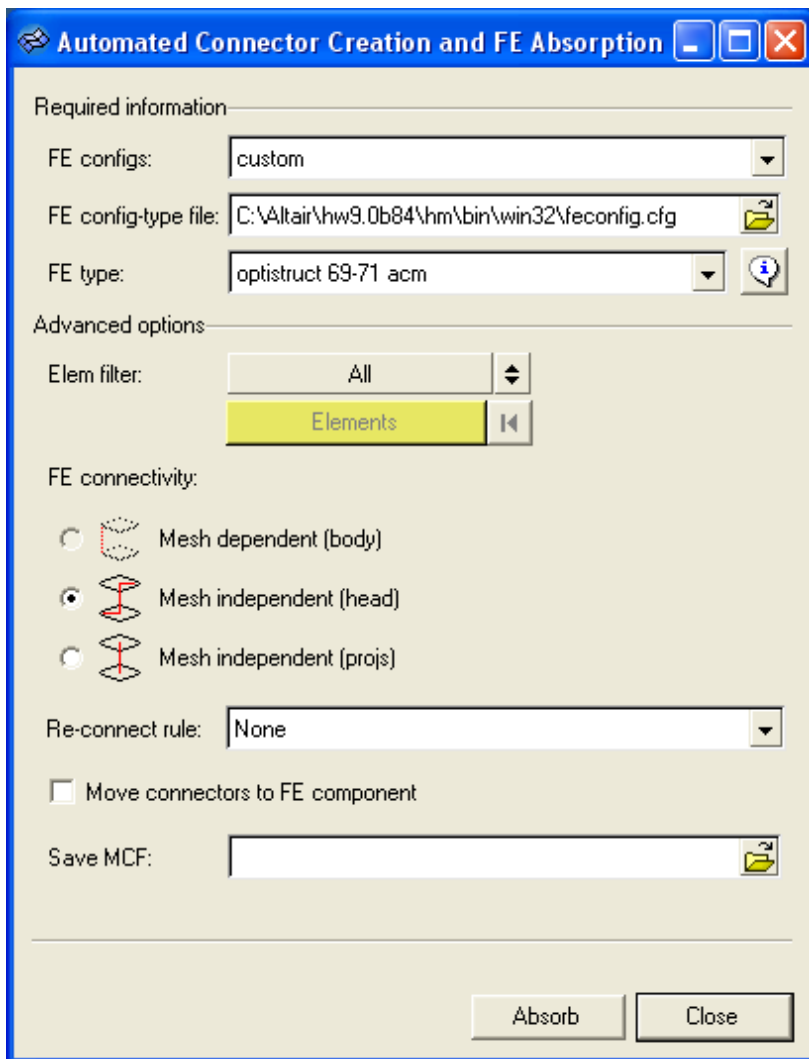
Step 23: Display only the assembly **assem_5** for elements and geometry.

1. Click the **Model** tab in the **tab** area. Or, go to the **View** menu and click **Model Browser**.
2. At the top of the browser window, verify that **Elements + Geometry** () is the current selection type.
3. Right-click **assem_5** and select **Isolate**.

Step 24: Create connectors from existing ACM welds.

Use the **fe absorb** panel to obtain connectors from the existing ACM welds (elements) in the component, **Con_Rear_Truss**.

1. Enter the **Automated Connector Creation and FE Absorption** dialog in one of the following ways:
 - From the **Pull-down** menu, select **Connectors**, then select **FE Absorb**
 - At the bottom of the **Connector Browser** click **FE Absorb GUI** ()



2. Set **FE Configs**: to **custom**.
3. Set **FE Type**: to **optistruct 69-71 acm**.
4. Toggle **Elem filter**: from **All** to **Select**.
5. Click the **Elem filter: Elements** selector twice.
A **HyperMesh** panel menu with an **elems** selector appears.
6. Select **elems >> by collector** and select the component, **Con_Rear_Truss**.
7. Click **proceed** to return to the dialog.
8. Activate the **Move connectors to FE component** option.
9. Click **Absorb** to absorb the elements into connectors.

Connectors are generated at the locations of the ACM welds. They are realized and are 2t connectors. Also, the connectors are organized into the **Con_Rear_Truss**, the same component to which the ACMs belong.

10. Click **Close** to close the dialog.

Exercise 2: Creating Area Connectors

In this tutorial, you will learn how to:

- Apply an adhesive connection to the left rails.

This exercise uses the model file, `frame_assembly1.hm`. Area connectors must be meshed in order to work properly. When the connector's *location* is existing FE mesh *elems*, the connector automatically gets meshed to match the elements chosen. However, after creating an area connector on *surfs*, *lines*, or along *nodes*, you must then use the automesh options (which display when you select one of these locations types) to create a mesh on the connector area.

area	Create and realize bolt connectors in a single process.
create	Create, but do not realize, area connectors.
realize	Create FE representations of previously-created area connectors.

Step 1: Retrieve and view the model file.

1. Open the model file, `frame_assembly1.hm`.
2. Take a few moments to observe the model using various visual options available in HyperMesh (rotation, zooming, etc.).
3. Go to the **Preferences** menu, click **User Profiles**.
4. Select **Radioss**, then **Bulk Data**.
5. Click **OK** to load the user profile.

Step 2: Load the **Connector Browser**.


1. Go to the **View** menu and click **Connector Browser**.
2. Review the layout of the **Connector Browser**.

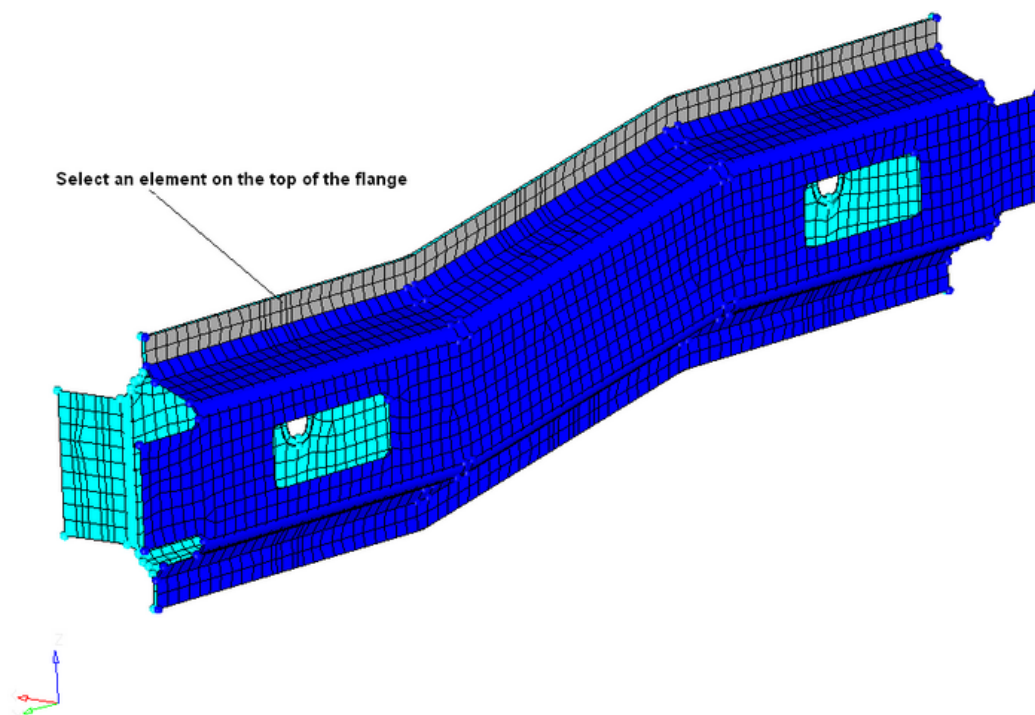
The **Connector Browser** allows users to view and manage the connectors in their model. The top portion of the browser is called the **component window** and it contains a listing of the components being connected with the connectors. The lower portion is called the **connector window** and it contains a list of the connectors in the model. The connectors will be grouped based on the type of connection.

Currently there are no components or connectors listed because there are no connectors in your model.

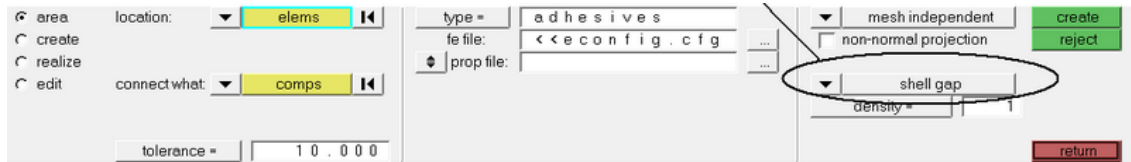
Step 3: Create an adhesive connection between component, **Left_Rail_1** and **Left_Rail_2** on the top flange.

1. From the **model browser**, display only the elements for components, **Left_Rail_1** and **Left_Rail_2**.
2. Zoom into an area displaying the two flanges and inspect the elements to be joined.

3. Create a new component by right-clicking in the **Model Browser** and select **Create >> Component**. Call it, **Left_Rail_Adhesive**. Ensure the new component is the current component.
4. Access the **area** panel in one of the following ways:
 - Right-click in the **Connector Browser** in the connector window and select **Create >> area**.
 - From the main menu, select the **1D** page, then select the **connectors** module, then select **area**.
 - At the bottom of the **Connector Browser** click **area** (). Access the **area** sub-panel.
5. Verify that **location:** is set to **elems**.
6. Pick one element on the top flange of **Left_Rail_1**. The element should be highlighted.
7. Click **elems** and select the **by face** option – the whole flange should be highlighted.



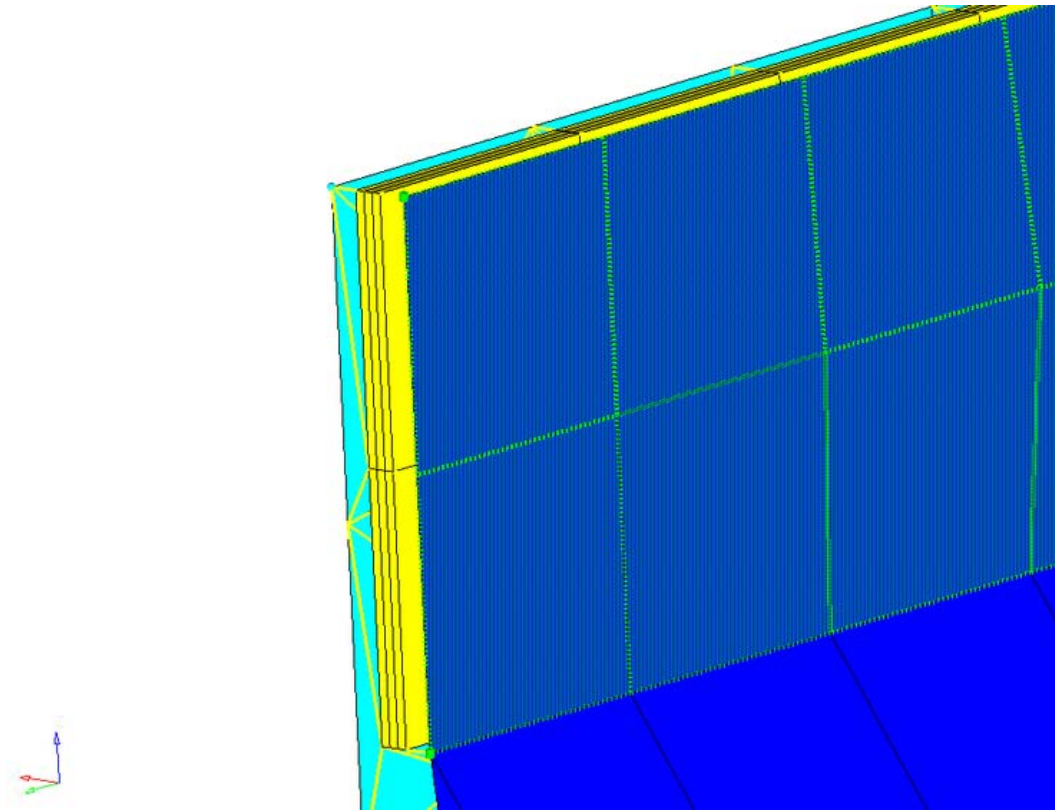
8. Next, select **comps** and pick **Left_Rail_1** and **Left_Rail_2** for **connect what**
9. Set **tolerance=** 10. The connector will connect any selected entities within this distance of itself.
10. Click **type=** and select **adhesives**.
11. Verify that the adhesive type is set to **shell gap** – this option projects directly to the shell component and takes no account of the thickness of the shell component.



12. Click **create**.

13. Inspect the new adhesive, note that an area connector has been created, and then click **reject**.

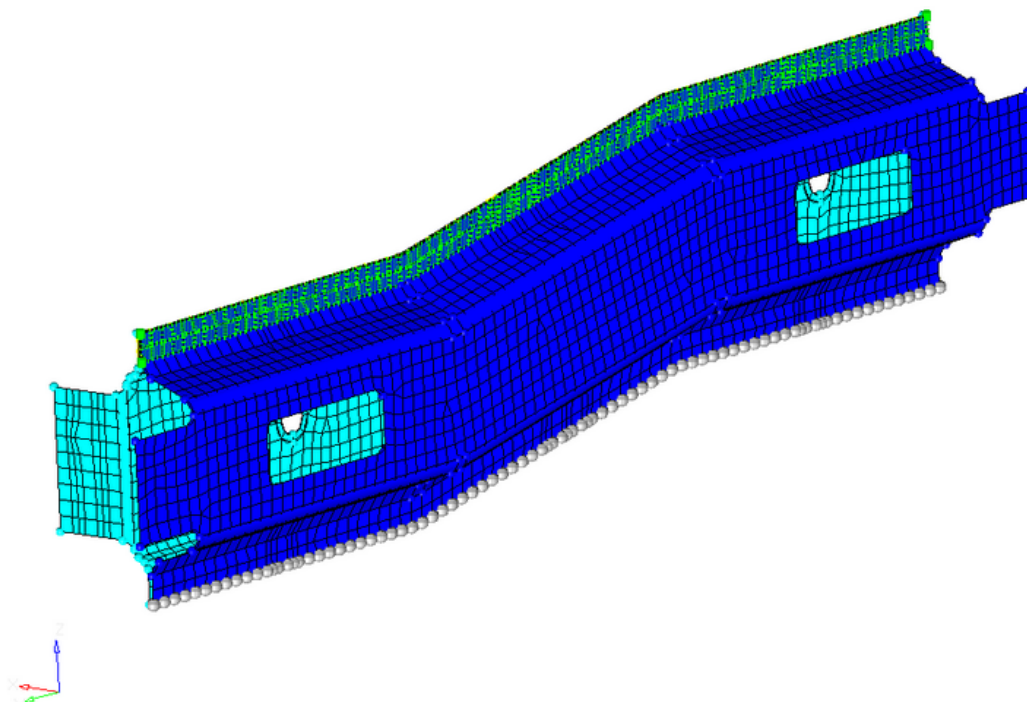
14. Repeat #5 – #12, and select the type to $(T1+T2)/2$ (instead of **shell gap**) and increase the **density**= 3. This type takes into account the thickness of each shell part and increases the number of hexas through thickness from 1 to 3.



For the other set of flanges you will manually create an area connector and mesh it accordingly.

Step 4: Create an adhesive connection between component, Left_Rail_1 and Left_Rail_2 on the bottom flange.

1. Verify that you are still in the **area** panel.
2. Select **nodes** for the **location**. Click **node list** and select by **path**. Select the row of nodes on the outer flange on part, **Left_Rail_1** by first selecting the left most node on the bottom flange of **Left_Rail_1** and then selecting the right most node on the bottom flange (see following image).



3. Define **width** = 10
4. Define **offset** = 3
5. Select **comps**, **Left_Rail_1** and **Left_Rail_2**.
6. Click **create**.

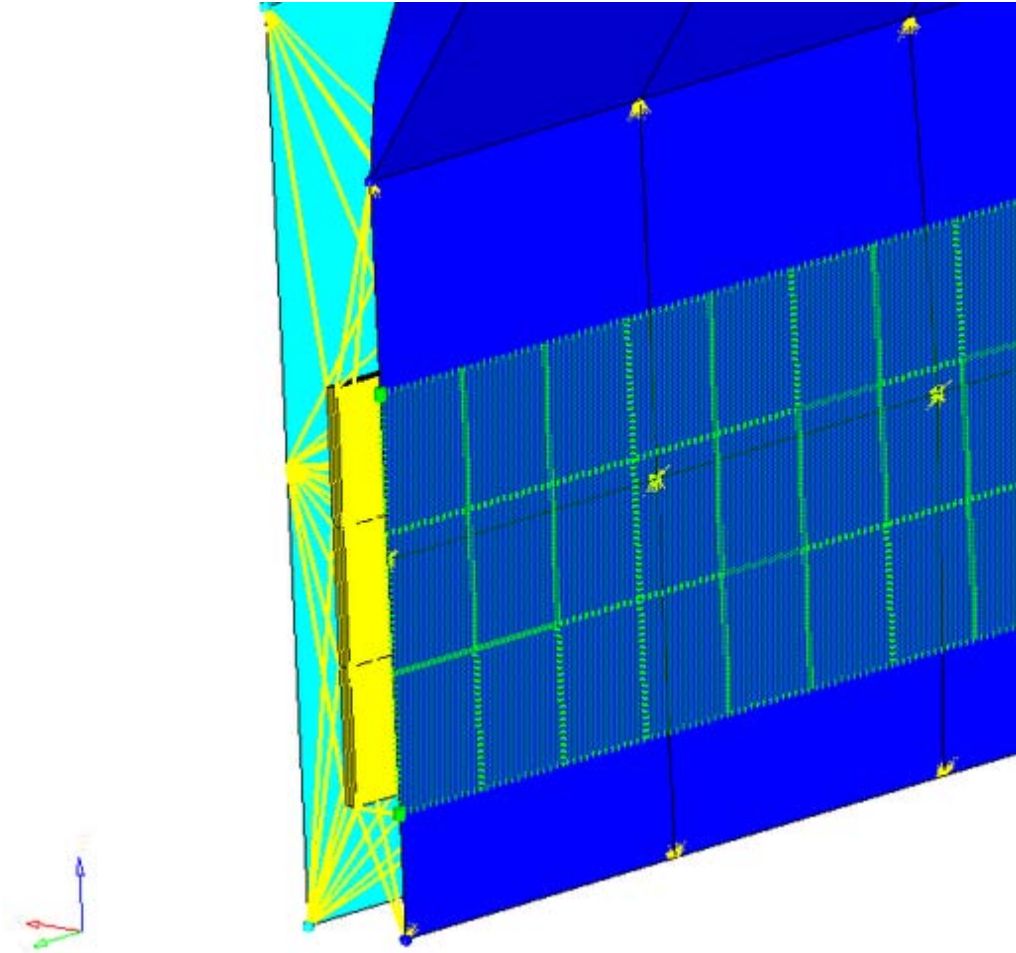
The default mesh size for these mesh independent area connectors (when choosing by nodes/lines/surfs) is 10. There is, however, an option to control the mesh size.

7. Go to the **edit** sub-panel.

8. Select the newly created area connector.
9. Select the **element size**= 3.
10. Click **mesh**.
- Note the connector unrealizes if there is a pre-existing mesh.
11. Access the **realize** sub-panel and pick the connector.
12. Change the **type** from **(T1+T2)/2** to **const_thickness** and enter 0.3.

13. Click **realize**.

Inspect the new adhesive created. Note when creating area connectors from elements it will automatically mesh the area connector using the current mesh. If the area connector is created from nodes, lines or surfaces then a manual mesh can be applied if the default is unsuitable from the **area** sub-panel.



14. Return to the main menu.

Exercise 3: Creating Bolt Connectors

In this tutorial, you will learn how to:

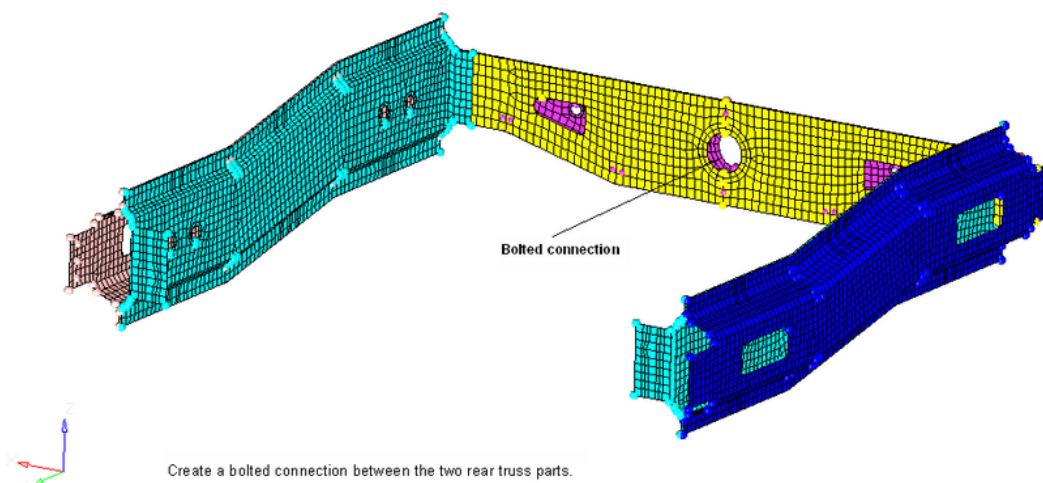
- Apply a bolted connection to the two rear trusses.

This exercise uses the model file, `frame_assembly2.hm`. The **bolt** panel creates connectors based on holes within the connected components, using spiders or washers at each end of an RBE connector. When the bolt panel is active, only bolt-type connectors display in the **graphics** area; graphics for other connector types are suppressed until you exit the panel. The **bolt** panel contains three sub-panels:

bolt	Create and realize bolt connectors in a single process.
create	Create, but do not realize, bolt connectors.
realize	Create FE representations of previously-created bolt connectors.




Step 1: Retrieve and view the model file.

1. Retrieve the model file, `frame_assembly2.hm`.
2. Take a few moments to observe the model using various visual options available in HyperMesh (rotation, zooming, etc.).
3. Go to the **Preferences** menu, click **User Profiles**.
4. Select **Radioss**, then **Bulk Data**.
5. Click **OK** to load the user profile.



Step 2: Display only the assembly, `assem_5` for elements and geometry.

1. Click the **Model** tab in the **tab** area. Or, go to the **View** menu and click **model browser**.

2. Select the **Model View** icon, , in the **model browser** and expand the **Assembly Hierarchy** tree.
3. Right-click the **entity selection** icon, , and select **Elements + Geometry** (). This turns on/off both elements and geometry when performing right-click **operations** in the **Model Browser**.
4. Right-click **assem_5** and select **Isolate**. This will only display the components that are in the **assem_5** assembly.
5. Set **Con_Rear_Truss** to be the **Current Component** by right-clicking it in the **Model Browser** and selecting **Make Current**.


Step 3: Load the **Connector Browser**.

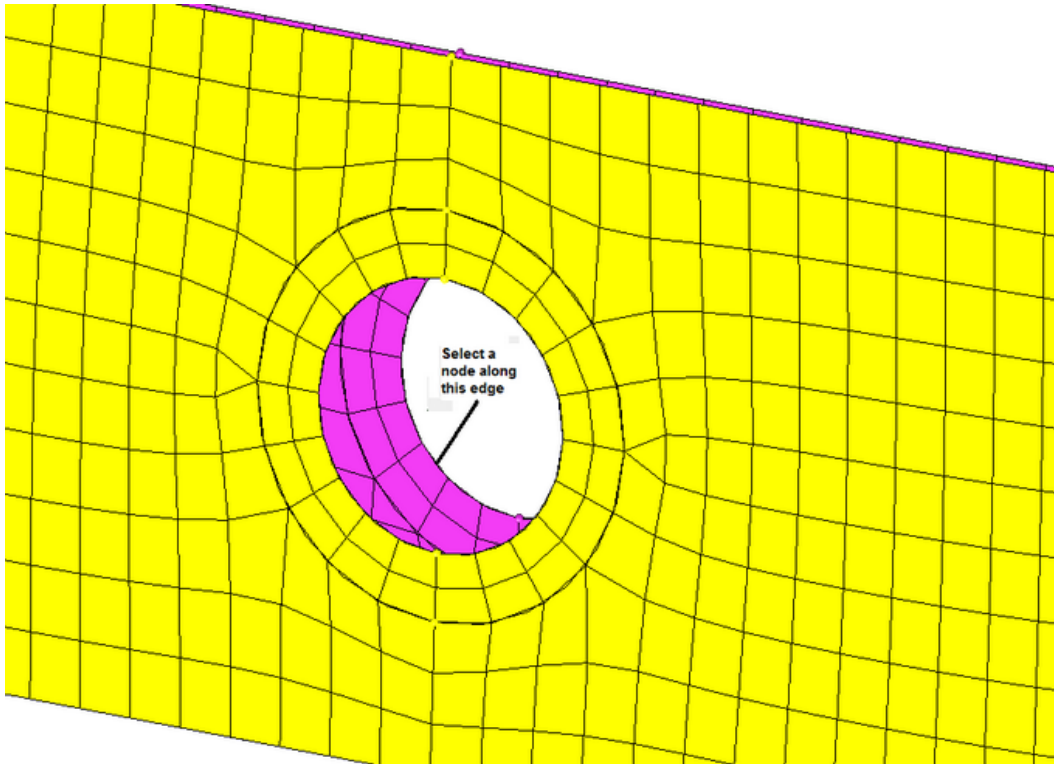
1. Go to the **View** menu and click **Connector Browser**.
2. Review the layout of the **Connector Browser**.

The **Connector Browser** allows users to view and manage the connectors in their model. The top portion of the browser is called the **component window** and it contains a listing of the components being connected with the connectors. The lower portion is called the **connector window** and it contains a list of the connectors in the model. The connectors will be grouped based on the type of connection.

Currently there are no components or connectors listed because there are no connectors in your model.

Step 4: Create a bolt connector.

1. Access the **bolt** panel in one of the following ways:
 - Right-click in the **Connector Browser** in the connector window and select **Create >> bolt**.
 - From the main menu, select the **1D** page, then select the **connectors** module, and then select **bolt**.
 - At the bottom of the **Connector Browser** click **bolt** ()
2. Verify that **location** is set to **nodes** and select the node on the edge of the hole in the component, **Rear_Truss_1**.



3. Pick the components to connect **Rear_Truss_1** and **Rear_Truss_2**.
4. In the **tolerance=** field, type 50.
The connector will connect any selected entities within this distance of itself.
5. Set **type=** to **bolt** (general).
Re-realizing the connector will allow you to see the different bolt types.
6. In the **max=** field, type to 60 to ensure that the diameter of the picked hole will be captured.
7. Click **create**. Ensure the display of the current component is turned on.
8. Click **return** to access the main menu.

