

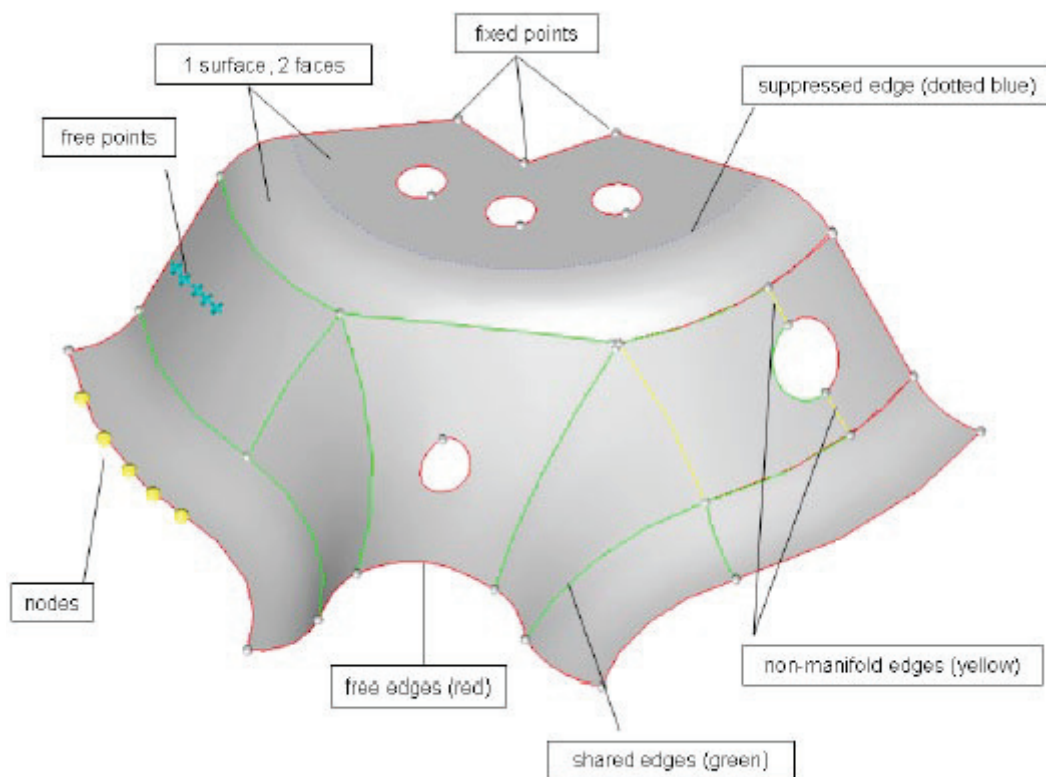
III

Geometry in HyperMesh

This chapter has been reviewed and has additional material added by Matthias Goelke.

3.1 HyperMesh Geometry Terminology

While dealing with geometry it is important to be familiar with the relevant HyperMesh terminology:



As mentioned in the previous chapters, your CAE project typically starts with the import of given CAD data e.g. CATIA, STEP, UG, IGES, SolidWorks, SolidThinking etc. (of course, you may create your CAD model in HyperMesh as well).

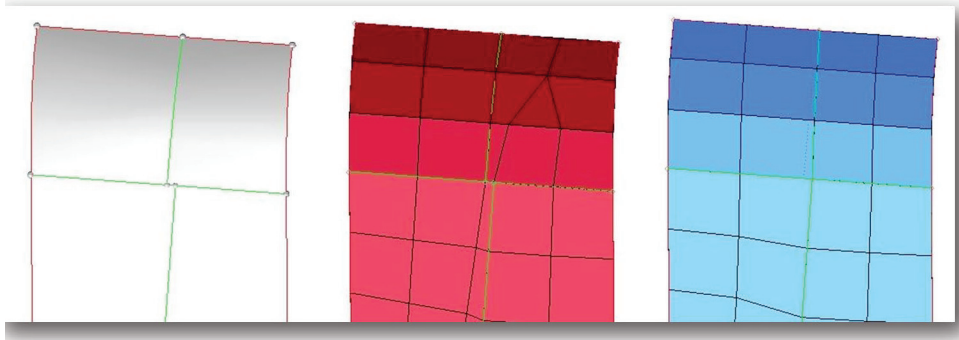
While the importation of data generally occurs with little error, there are issues that can occur, and as such, HyperMesh offers a wide variety of tools to remedy these geometric issues (this is one of the many reasons why HyperMesh is used in so many places).

Some of the issues described below do exist because when designers create CAD geometry, their priorities are different from those of analysts trying to use the data. For a designer, a single smooth surface is typically split into smaller patches.



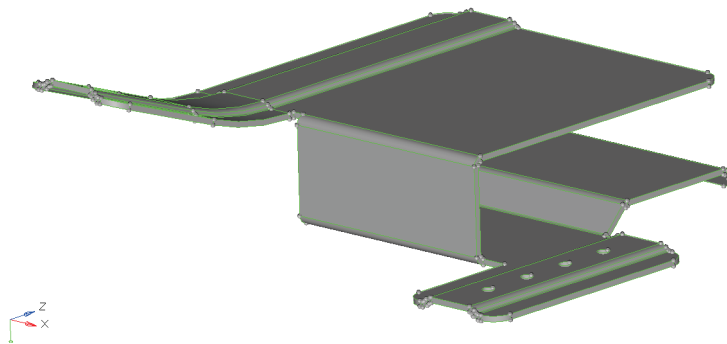
Some resulting geometry issues:

- Surfaces are not stitched together (i.e. there is a gap between surfaces)
- Very small surfaces are squeezed between regular surfaces
- The juncture between two surfaces often contains gaps, overlaps, or other misalignments

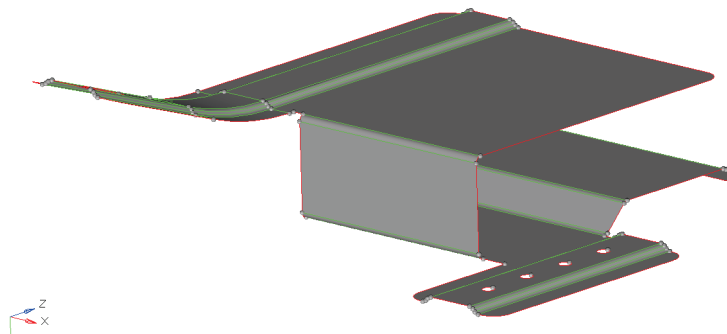


Left: CAD with “jumps” results in irregularly shaped elements (middle). Right: improved CAD with regular mesh

- The geometry is a thin-walled volume structure (i.e. instead of building a complex an exhaustive 3D mesh, a corresponding mid-surface model meshed with 2D elements would be far better



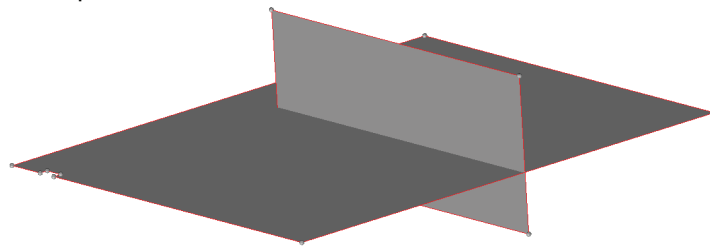
Original thin walled structure



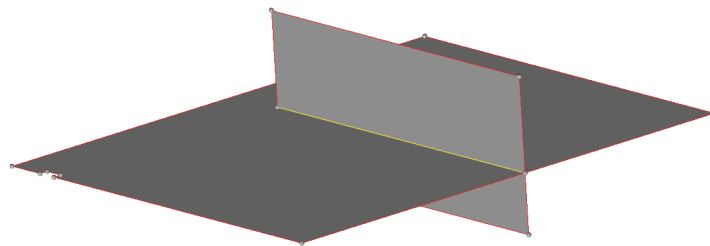
Midsurface representation of the thin walled structure



- Surfaces penetrate each other (such as at a t-connection) but don't "feel" each other

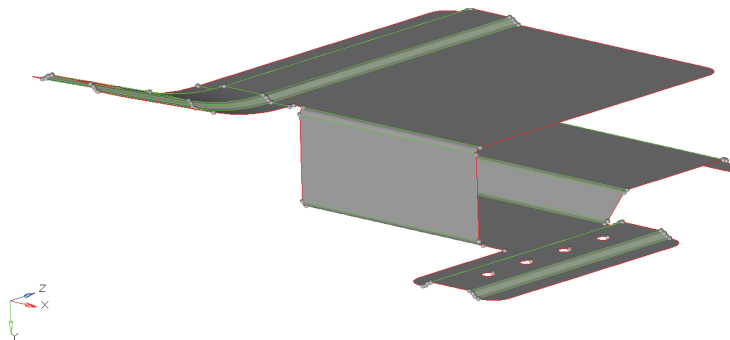


Original Geometry

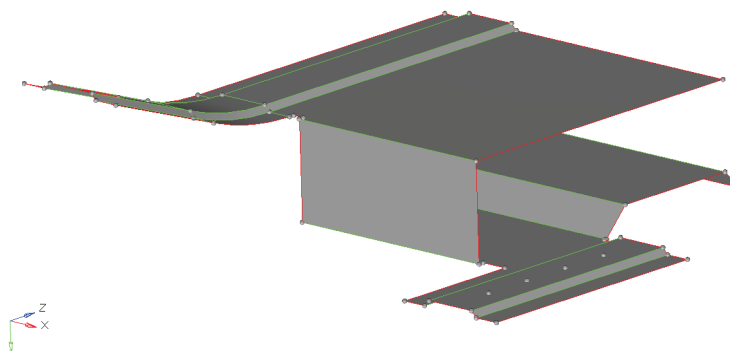


Geometry fixed so that there is proper connectivity

- Geometry is much too detailed (e.g. tiny fillets which are not needed for the analysis)



Original geometry, note the small fillets



Simplified geometry where small fillets are replaced by a sharp edge

- And many others ...

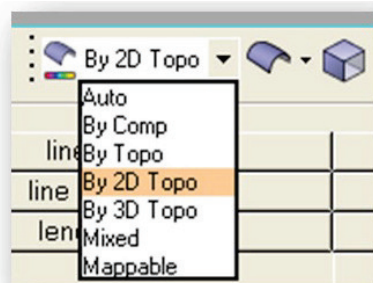
3.2 Geometry Cleanup

All of the issues defined in the previous section typically demand what is called Geometry Cleanup.

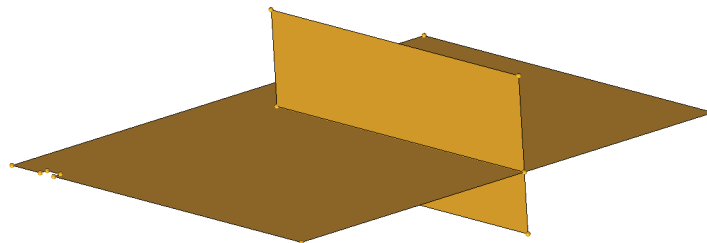
Topology Repair: Strategy

Below is a general strategy that can be followed to perform the topology repair. This is a generalized strategy which may need to be changed to suit the needs of your model, but it provides a good starting point to perform the topology repair.

1. 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.
2. 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.
3. Use topology display tools to determine what needs to be fixed. For instance, to display the topology of 2D geometry set the selector to "By 2D Topo"

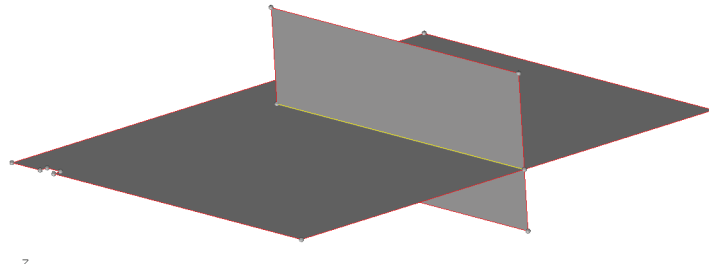


- Visualization mode: By Comp (which takes the color of the component)

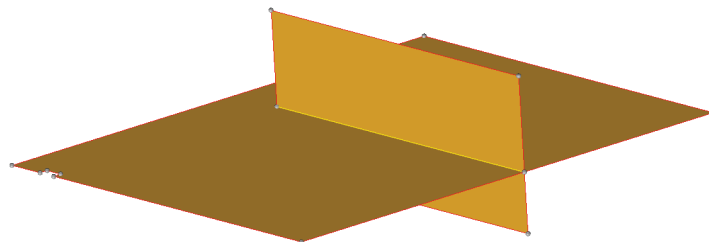




- Visualization mode: By 2D Topo



- Visualization mode: Mixed which takes the component color and add topology information



4. Find duplicate surfaces and delete them.
 - To delete duplicate surfaces, from the menu bar select **Geometry > Defeature > Duplicates**.
5. Use **equivalence** to combine as many free edge pairs as possible.
 - Visually verify no surfaces were collapsed with this function.
6. Use **toggle** to combine any remaining edges.
 - Use replace if more control is needed.
7. Use **filler surface** to fill in any missing surfaces.
8. The **equivalence**, **toggle**, and **filler surface** can be accessed within the **Quick Edit** panel.
 - To access the **Quick Edit** panel, from the menu bar select **Geometry > Quick Edit**.

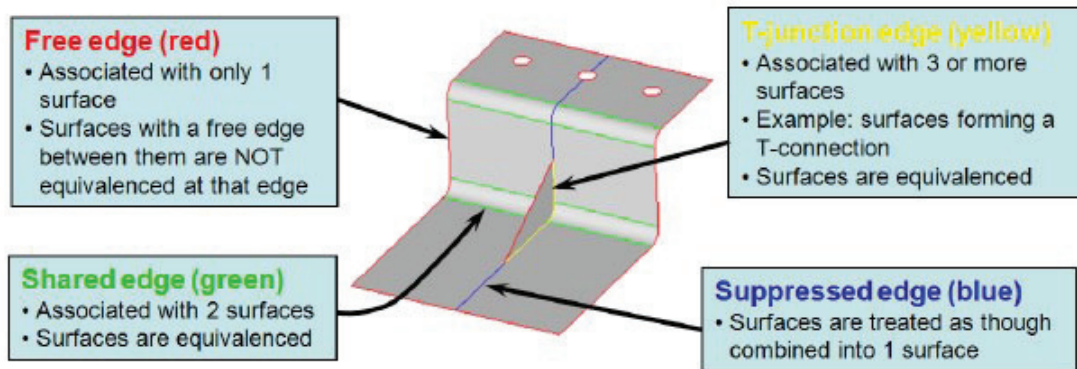
split surf-node:	node	node	↑	adjust/set density:	line(s)	line(s)	reject
split surf-line:	node	line	↓	replace point:	point(s)	retain	
washer split:	line(s)	offset value:	0 . 1 0 0	add/remove point:	point(s)		
unsplit surf:	line(s)			add point on line:	line(s)	no. of points:	1
toggle edge:	line(s)	tolerance:	0 . 0 1 0	release point:	point(s)		
filler surf:	line(s)			project point:	point(s)	line	↑
delete surf:	surf(s)			trim-intersect:	node	node	return

Topology Repair: Tools and Panels

The perimeter of a surface is defined by edges. There are four types of surface edges:

1. Free edges
2. Shared edges
3. Suppressed edges
4. Non-manifold edges

Surface edges are different from lines and are sometimes handled differently for certain HyperMesh operations. The connectivity of surface edges constitutes the geometric topology. Below the four types of surface edges which represents the geometric topology is described (Note: the shown model is displayed via the “2DTopo” mode in HyperMesh).



Free Edges

A free edge is an edge that is owned by only one surface. Free edges are colored red by default.

On a clean 2-D model consisting of surfaces, free edges appear only along the outer perimeter of the part and around any interior holes. **Note: Free edges that appear between two adjacent surfaces indicate the existence of a gap between the two surfaces. The automeshing will leave a gap in the mesh wherever there is a gap between two surfaces.**

Shared Edges

A shared edge is an edge that is owned, or shared, by two adjacent surfaces. Shared edges are colored green by default.

When the edge between two surfaces is a shared edge (this is what you typically want to have), there is no gap or overlap between the two surfaces - they are geometrically continuous. The automeshing always places seed nodes along the length of a shared edge and will produce a continuous mesh without any gaps along that edge. The automeshing will not construct any individual elements that cross over a shared edge.

Suppressed Edges

A suppressed edge is shared by two surfaces but it is ignored by the automeshing. Suppressed edges are colored blue by default.

Like a shared edge, a suppressed edge indicates geometric continuity between two surfaces but, unlike a shared edge, the automeshing will mesh across a suppressed edge as if were not even there. The automeshing does not place seed nodes along the length of a suppressed edge and, consequently, individual elements will span across it. By suppressing undesirable edges you are effectively combining surfaces into larger logical meshable regions.

Non-manifold Edges

A non-manifold edge is owned by three or more surfaces. Non-manifold edges are colored yellow by default.

They typically occur at “T” intersections between surfaces or when 2 or more duplicate surfaces exist. The automeshing always places seed nodes along their length and will produce a continuous mesh



without any gaps along that edge. The automesher will not construct any individual elements that cross over a T-joint edge. These edges cannot be suppressed.

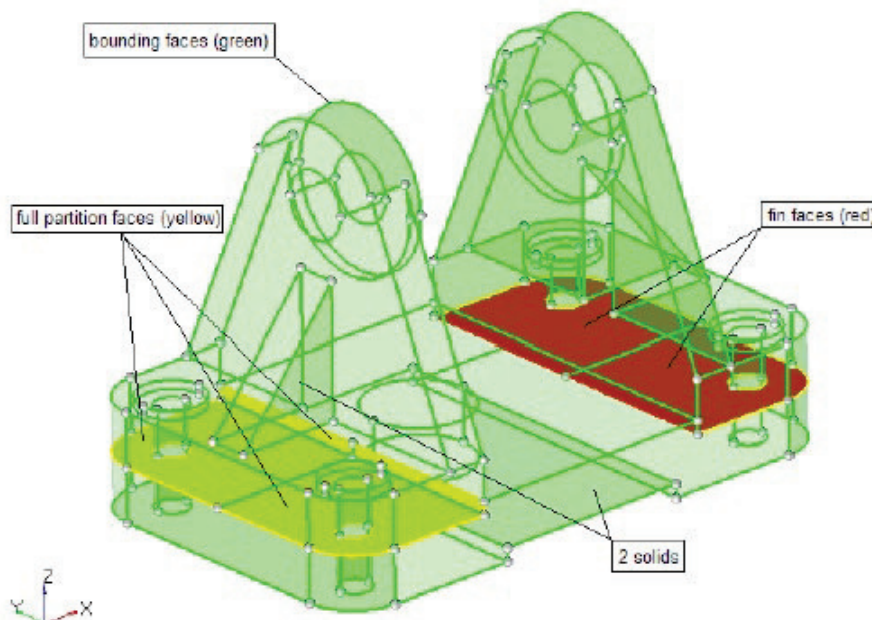
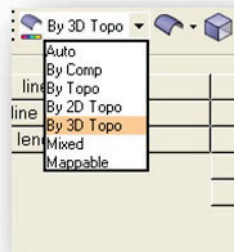
Solids

A solid is a closed volume of surfaces that can take any shape. Solids are three-dimensional entities that can be used in automatic tetra and solid meshing. Its color is determined by the component collector to which it belongs.

The surfaces defining a solid can belong to multiple component collectors. The display of a solid and its bounding surfaces are controlled only by the component collector to which the solid belongs.

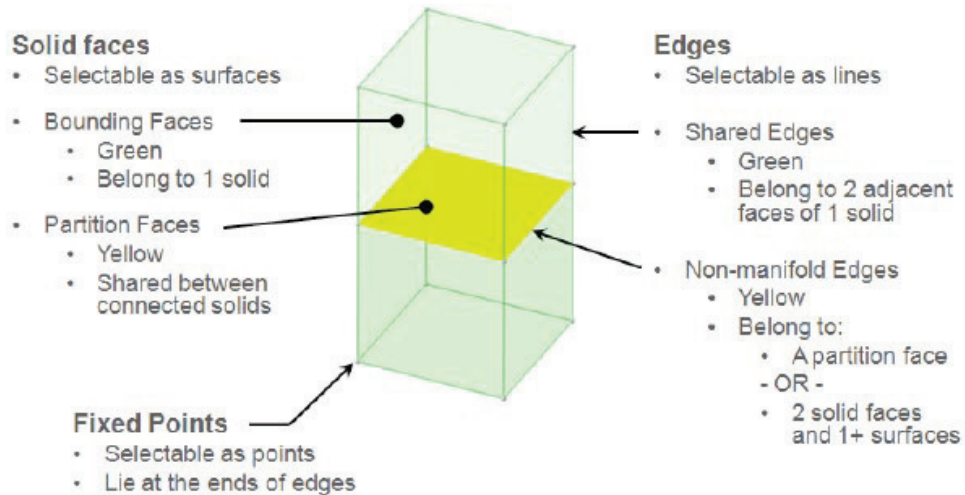
Below is an image of solid topology as well as a description of the three types of surfaces which define the topology of a solid.

To activate the 3D topology mode view please activate the corresponding setting





- Example: 2 connected solids in topology display



Bounding Faces

A bounding face is a surface that defines the outer boundary of a single solid. Bounding faces are shaded green by default.

A bounding face is unique and is not shared with any other solid. A single solid volume is defined entirely by bounding faces.

Fin Faces

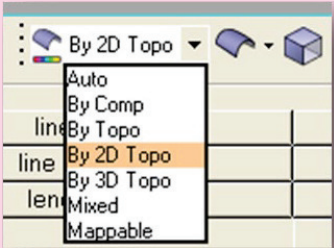
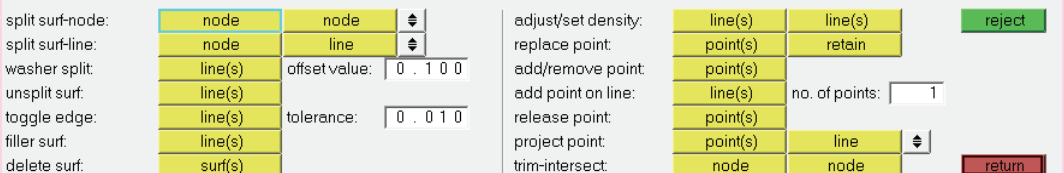
A fin face is a surface that has the same solid on all sides i.e. it acts as a fin inside of a single solid. Fin faces are shaded red by default.

A fin face can be created when manually merging solids or when creating solids with internal fin surfaces.

Full Partition Faces

A full partition face is a surface that defines a shared boundary between one or more solids. Full partition faces are shaded yellow by default.

A full partition face can be created when splitting a solid or when using Boolean operations to join multiple solids at shared or intersecting locations.

What you need to know or remember	
Green edges - 2 surfaces are stitched together; the FE mesh will be linked (compatible), its nodes will line up with the green edge.	
Red edges - indicates free surface edges. Red edges inside the geometry tell you that the surfaces are not stitched together (gap); the FE mesh will NOT be linked (not compatible).	
Yellow edges — minimum of 3 edges are stitched together; the FE mesh will be compatible.	
Blue edges — Suppressed green edge. Surfaces are “melted” together. In other words, the mesher does not see this edge and elements are placed across it.	
How to visualise the edge “colors”?	
	
Display is controlled in the Visualization Tool Bar by activating for instance “By 2D Topo” (surfaces turn into grey, edges are colored respectively) or “Mixed” (surfaces are displayed in their original color (reminder: surface color is controlled in the Model Browser), edges are colored respectively).	
Panels to be used: <ul style="list-style-type: none"> • Toggle surfaces (combining, stitching) • Trimming surfaces (splitting) • Suppressing combined edges 	
Geometry > Quick Edit opens up a very comprehensive panel which allows you (among many other options) to execute the above listed tasks.	
	

The before mentioned panels describe just a very minor fraction of HyperMesh’s geometry cleanup functionalities.

Once you feel more comfortable with the process, you will automatically explore and learn more about other techniques.

3.3 Geometry Creation and Editing

There are many different ways to create geometry in HyperMesh which include importing geometry from external CAD models, as well as creating new geometry from scratch. The methods used to create a particular geometry depend on both the entities available for input and the level of detail required.



The following is a list of the geometric entities which can be created and edited within HyperMesh:

- Nodes
- Free Points
- Fixed Points
- Lines
- Faces
- Surfaces
- Solids

For each of these entities, we will investigate how they can be created.

Nodes

A node is the most basic finite element entity. A node represents a physical position on the structure being modeled and is used by an element entity to define the location and shape of that element. It is also used as temporary input to create geometric entities.

A node may contain a pointer to other geometric entities and can be associated directly to them. For example, for a node to be translated along a surface, it must first be associated to the surface.

A node is displayed as a small circle or sphere, depending on the mesh graphics mode. Its color is always yellow.

Nodes are created using the menu bar by selecting **Geometry > Create > Nodes** and then selecting a method to create the nodes with.

Free Points

A free point is a zero-dimensional geometry entity (for more information see: HyperMesh > HyperMesh Entities & Solver Interfaces > Collectors and Collected Entities in the help) in space that is not associated with a surface.

It is displayed as a small "x" and its color is determined by the component collector to which it belongs. These types of points are typically used for weld locations and connectors.

Free points are created using the menu bar by selecting **Geometry > Create > Free Points** and then selecting a method to create the free points with.

Fixed Points

A fixed point is a zero-dimensional geometry entity in space that is associated with a surface. Its color is determined by the surface to which it is associated.

It is displayed as a small "o". The automeshing places an FE node at each fixed point on the surface being meshed. These types of points are typically used for weld locations and connectors.

Fixed points are created using the menu bar by selecting **Geometry > Create > Fixed Points** and then selecting a method to create the fixed points with.

Lines

A line represents a curve in space that is not attached to any surface or solid. A line is a one-dimensional geometric entity. The color of a line is determined by the component collector to which it belongs.

A line can be composed of one or more line types. Each line type in a line is referred to as a segment. The end point of each line segment is connected to the first point of the next segment. A joint



is the common point between two line segments. Line segments are maintained as a single line entity, so operations performed on the line affect each segment of the line. In general, HyperMesh automatically uses the appropriate number and type of line segments to represent the geometry.

It is important to realize that lines are different from surface edges and are sometimes handled differently for certain HyperMesh operations.

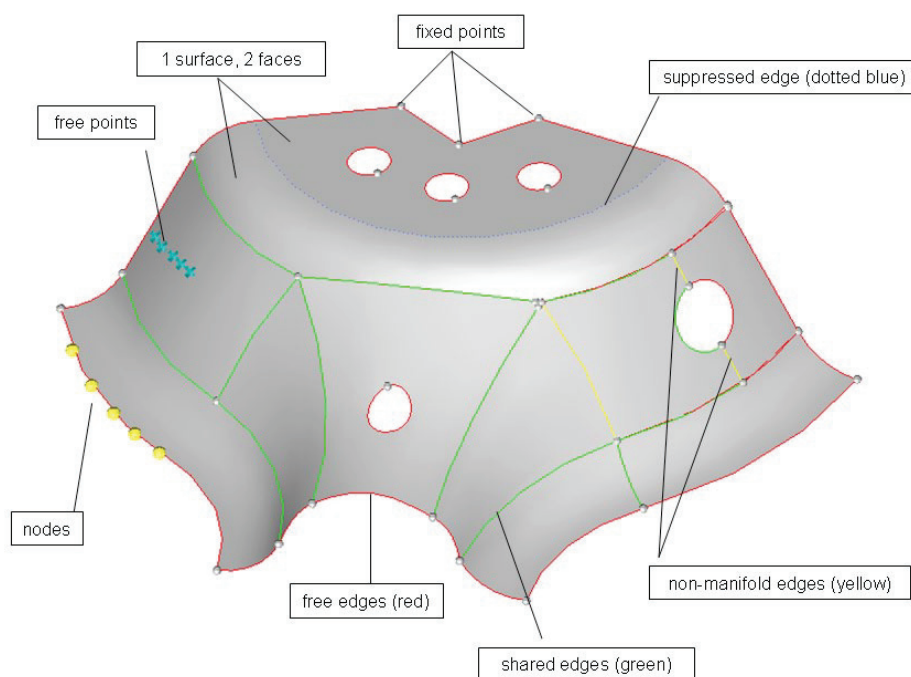
Lines are created using the menu bar by selecting **Geometry > Create > Lines** and then selecting a method to create the lines with.

Surfaces

A surface represents the geometry associated with a physical part. A surface is a two-dimensional geometric entity that may be used in automatic mesh generation. Its color is determined by the component collector to which it belongs.

A surface is comprised of one or more faces. Each face contains a mathematical surface and edges to trim the surface, if required. When a surface has several faces, HyperMesh maintains all of the faces as a single surface entity. Operations performed on the surface affect all the faces that comprise the surface. In general, HyperMesh automatically uses the appropriate number of and type of surface faces to represent the geometry.

Surfaces are created using the menu bar by selecting **Geometry > Create > Surfaces** and then selecting a method to create the surfaces with.



Solids

A solid is a closed volume of surfaces that can take any shape. Solids are three-dimensional entities that can be used in automatic tetra and solid meshing. Its color is determined by the component collector to which it belongs.


The surfaces defining a solid can belong to multiple component collectors. The display of a solid and

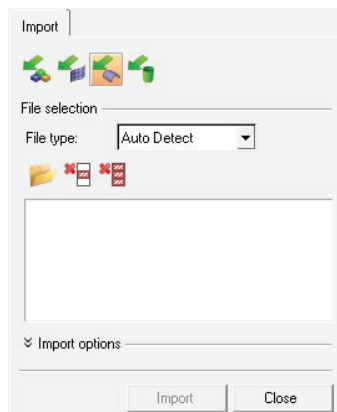
its bounding surfaces are controlled only by the component collector to which the solid belongs.

Solids are created using the menu bar by selecting **Geometry > Create > Solids** and then selecting a method to create the solids with.

Note & reminder: The newly created geometry will be placed /stored in the currently active component collector. Check the Model Browser for the current component collector.

3.4 Importing CAD Geometry

To import geometry, the **Import Browser**, accessible through the **Import Geometry** icon , is used.



Using the **Import Browser**, 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)



In addition, HyperMesh also supports the following CAD packages for Geometry Import:

- **ACIS**
- **DXF**
- **JT**
- **Parasolid**
- **PDGS**
- **VDAFS**

Advanced Import Options

The **cleanup tolerance** is used to determine if two surface edges are the same and if two surface vertices are the same. The **cleanup tol** toggle controls the following items:

- if two surface edges are close enough to be automatically combined into a shared edge (green edges)
- if a surface is degenerate and should be removed.

If you use the **Automatic** cleanup tolerance option, the complexity of the surface and edge geometries are taken into account and a tolerance to maximize the shared edges (green edges) is selected. The **automatic** cleanup tolerance value defaults to 100 times what is used internally by the translator.

If you want to specify a different value, use the **Manual** cleanup tolerance option, which must be greater than the default value. The translator modifies data only if the data stays within the original data tolerance. Increasing the tolerance can cause serious problems. When this value is set, any features equal to or less than the tolerance are eliminated. The translator does not include any edge less than the tolerance long; if there are edges present that are important to the surface, that surface will be distorted, or will fail to trim properly. Surfaces smaller than the tolerance may not be imported. If the file you have read has many very short edges, it may be worthwhile to reread the file using a larger tolerance. The same holds true if surfaces appear to be “inside out” when surface lines are displayed. The tolerance value should not be set to a value greater than the node tolerance (set in the **Options** panel) to be used for your element mesh. The Options panel is accessed through the menu bar by selecting **Preferences > Geometry Options**.

If you are reading a Catia file, you may need to override the tolerance in the file; in our experience, the tolerance in the file is almost always too small (by at least an order of magnitude).

- The **Automatic** option takes the complexity of the surfaces and edge geometries into account and a tolerance to maximize shared edges (green edges) is selected.
 - The **Manual** option allows you to set a specific tolerance in the field.
1. The **Import blanked (no show) components** option allows you to control if blanked components in the IGES translator will be imported into HyperMesh, as well as components containing “NO SHOW” entities from the Catia translator.
 2. Place a check in the **Name components by layer** option to activate this option. This option is valid for Catia V4 and Catia V5.
 - For Catia V4, the option is enabled by default, and can't be disabled.
 - For Catia V5, the option is disabled by default and can be enabled. If this option is enabled, Catia objects from the same layer are grouped into the same component.