Geometry Introduction

Importing and Repairing CAD

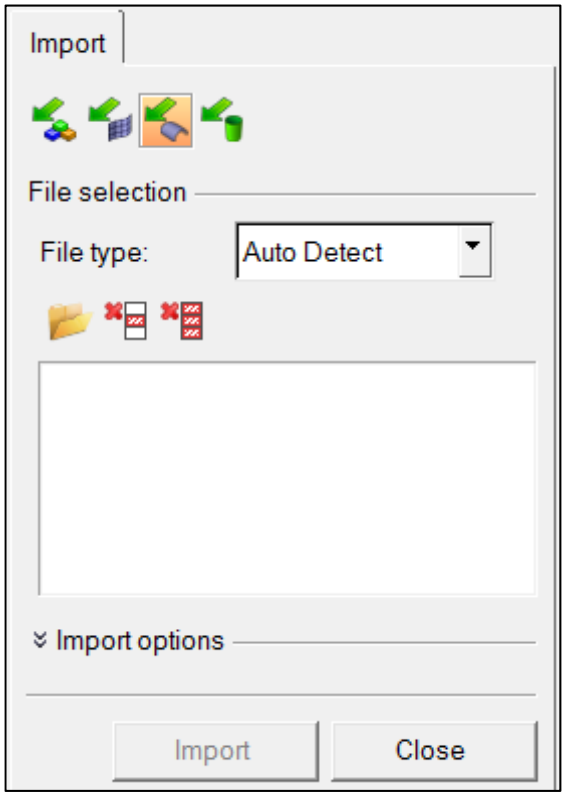
Generating Midsurface

Simplifying Geometry

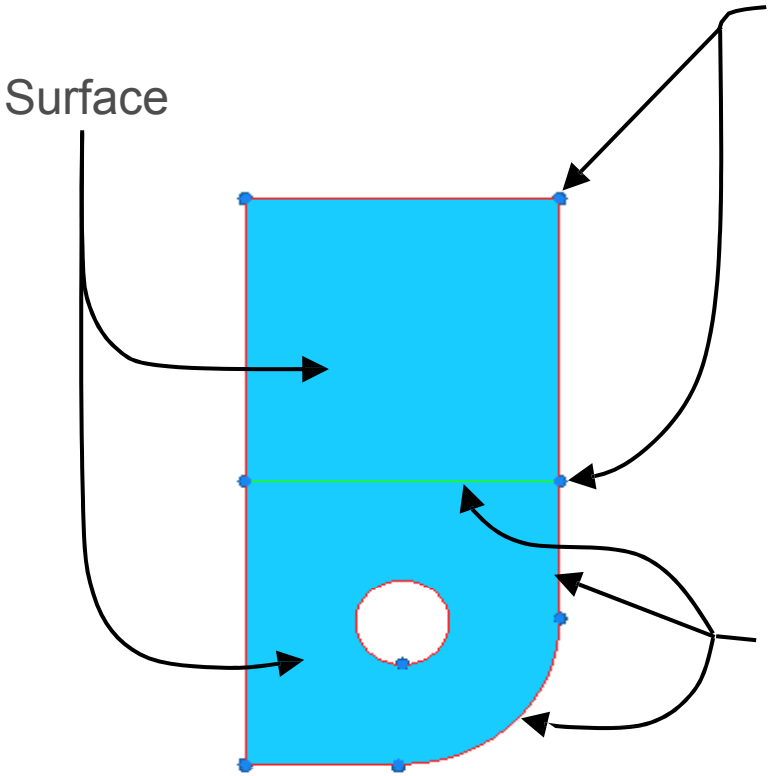
Refining Topology to Achieve a Quality Mesh

Topology Repair: Importing Geometry

- Import geometry data via:
  - ***Files*** > ***Import*** > ***Geometry*** drop-down menu
  - ***Toolbar*** >
- Common types of geometry files supported:
  - **Unigraphics** (**NX2, NX3, NX4, NX5)**
    - UG Part Browser
- Import of *.prt files
    - Requires an installation of Unigraphics to be available
  - **CATIA (V4 & V5)**
    - import of *.model files
    - CATIA V5 license required to import V5 files
  - **Pro/ENGINEER (Wildfire 2.0 & 3.0)**
    - import of *.prt and *.asm files
  - **IGES**
    - Import of *.igs / *.iges files
  - **STEP**
    - import of *.stp files



Topology Repair: Surface Definitions



Fixed point ("Surface vertex")

- Point associated with a surface
- Cannot be moved off the surface
- Can lie on a surface edge or the "interior" of a surface
- Separates surface edges from each other
- Forces a node to be placed at that location during meshing

Surface edge

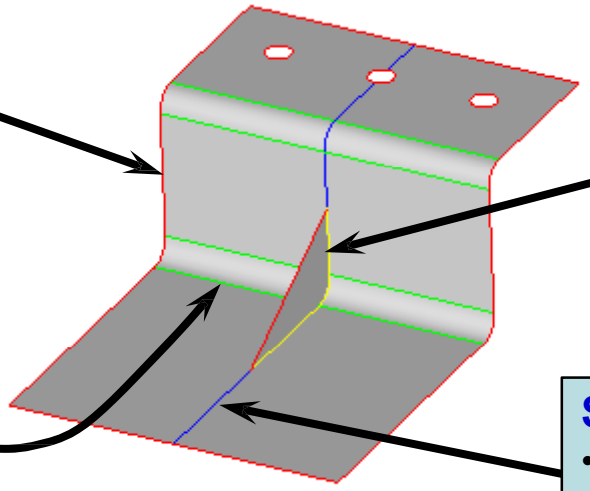
- Line associated with a surface
- Defines a surface's boundary
- Cannot be moved off the surface
- Has a fixed point on both ends

Automeshing: What is “topology”?

- Topology” is how surfaces connect to adjacent surfaces of a part
  - Surface connectivity is controlled by the associated surface edges
  - If a surface edge is associated with more than 1 surface, those surfaces are considered to be connected (“equivalenced”)
  - Surface edges are categorized, named, and colored according to the number of associated surfaces:

Free edge (red)
- Associated with only 1 surface
- Surfaces with a free edge between them are NOT equivalenced at that edge

T-junction edge (yellow)
- Associated with 3 or more surfaces
- Example: surfaces forming a T-connection
- Surfaces are equivalenced



Shared edge (green)
- Associated with 2 surfaces
- Surfaces are equivalenced

Suppressed edge (blue)
- Surfaces are treated as though combined into 1 surface

Topology Repair: Viewing Topology

Topology display mode is default for some panels (w/ Auto ON)

- ***surface edit, quick edit, point edit, edge edit, autocleanup,*** and ***automesh***

Can also be accessed via *geometry visualization type*

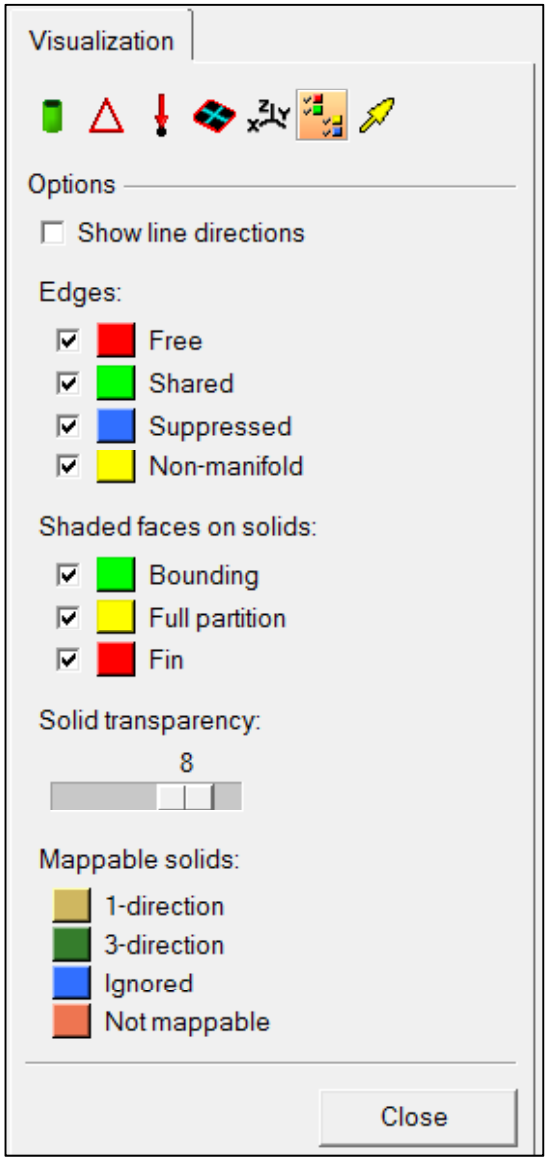
Auto	Default (topology display in only default panels mentioned above)
By Comp	Always in component color mode
By Topo	Always in topology display mode
By 2D Topo	Displays only 2D geometry in topology display mode
By 3D Topo	Displays only 3D geometry in topology display mode
Mixed	Displays 2D and 3D geometry in topology display mode
Mappable	Displays the solid entities in the various mappable states

Topology Repair: Viewing Topology

Toolbar →  ***Visualization*** **tab**

controls display of:

- Visibility of free, shared, t-junctions, and suppressed edges
- Level of surface transparency
- Solids Mappability



Topology Repair: What is it?

Altair University

HyperMesh will attempt to properly clean up surfaces during import

- Some types of geometry files have surface connectivity information which helps HyperMesh. Typically native geometry files like Catia, UG, ProE, etc.
- Geometry usually imports cleanly

Topology Repair consists of correcting connectivity errors between adjacent surfaces

- Possible errors include:
  - Unconnected adjacent surfaces
  - Duplicate surfaces
  - Missing surfaces

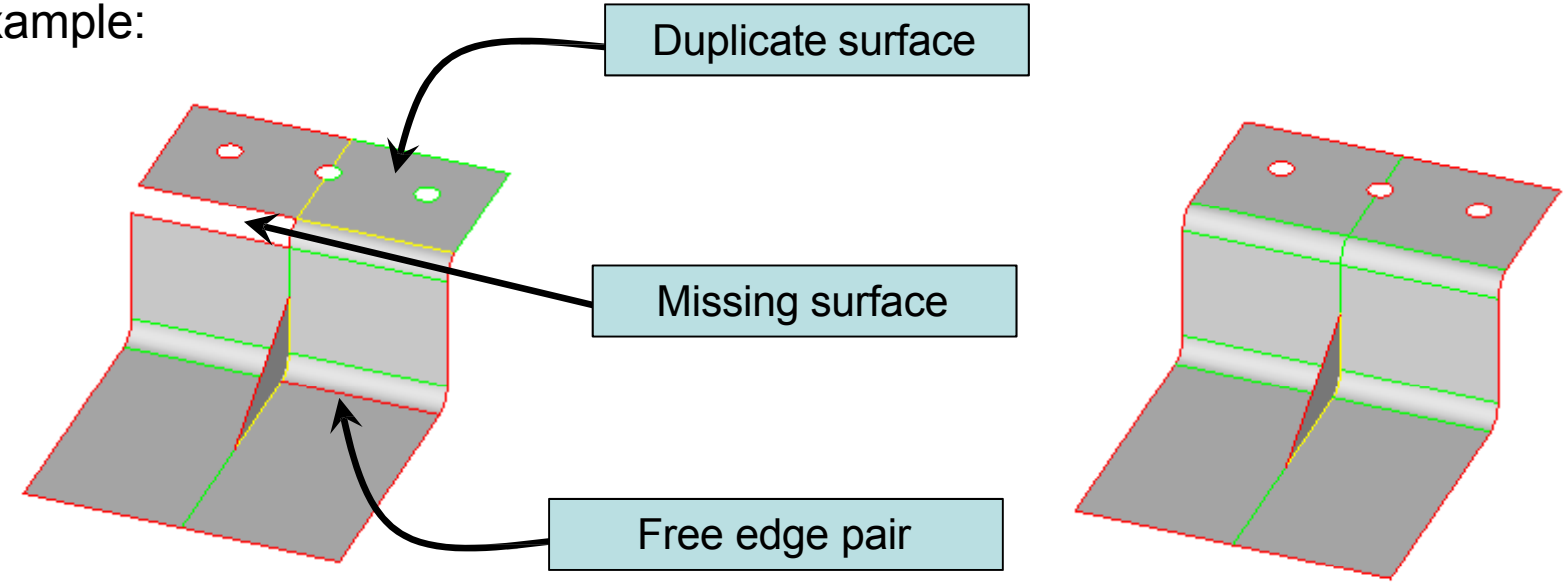
- **The Goal of Topology Repair:** Restore the surface data to a perfectly clean representation of the part

Topology Repair: Process

General process is to:

- Figure out what the ideal surface connectivity of the part should be
- Observe the current display of topology colors (free, shared, t-junction)
  - Figure out what is causing the topology to be displayed this way
- Use the tools in HyperMesh that get the connectivity from what it is to what it should be as quickly and efficiently as possible

Example:



Topology Repair: Tools

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 a shared edges

Defeature Panel

- ***Duplicates –*** Identify and delete duplicate surfaces within a user specified tolerance

Surfaces Panel

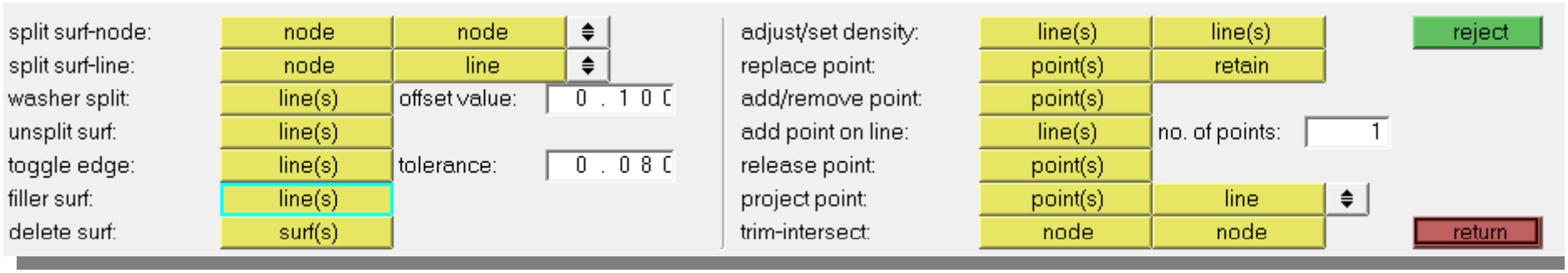
- ***Spline / filler –*** Select lines / surface edges to recreate any missing surfaces

Topology Repair: Tools

Quick Edit Panel

- Has a number of tools found in other panels
- Focused on tools with minimal user input for rapid editing

- ***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 model size & scale to determine an appropriate global element size

Set a cleanup tolerance based on the determined global element size

- Set appropriate value in ***options, geom cleanup***, and ***automesh : cleanup***
- Cleanup tolerance specifies the largest gap size to be closed by topology functions
- Tolerances > 15-20% of global element size can cause mesh distortions
- Can change value multiple times for work on various areas of the model

Use topology display tools to decide what needs to be cleaned

Use *equivalence* to combine as many free edge pairs as possible

- Make sure surfaces are not collapsed in undesirable manner

Use *toggle* to combine any remaining free edge pairs, 1 by 1

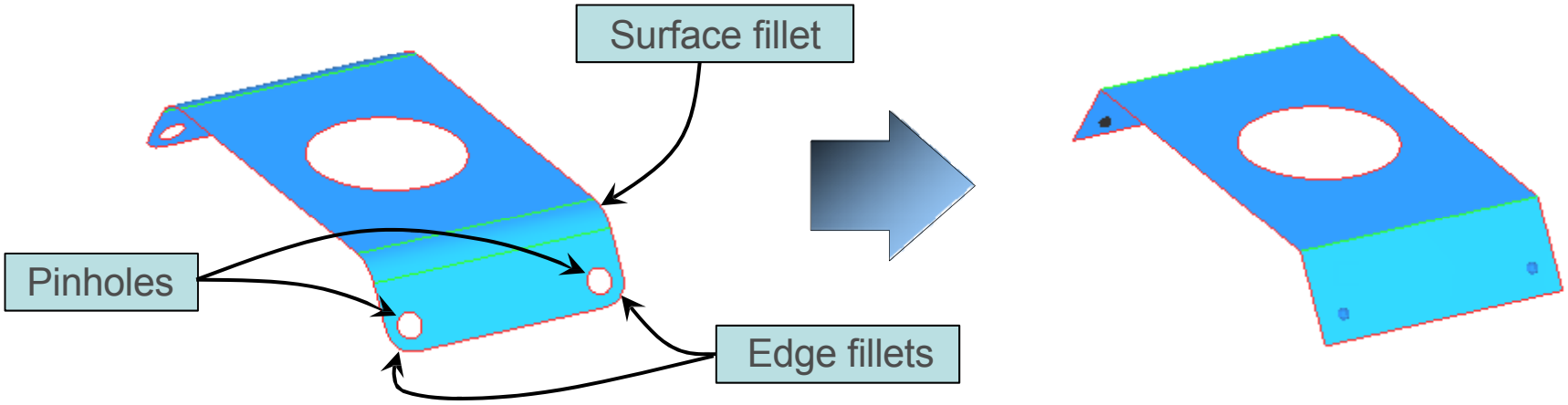
- use ***replace*** function if more control is needed

Use *find duplicates* to check for any duplicate surfaces and delete them

Use *filler surface* to recreate any missing surfaces

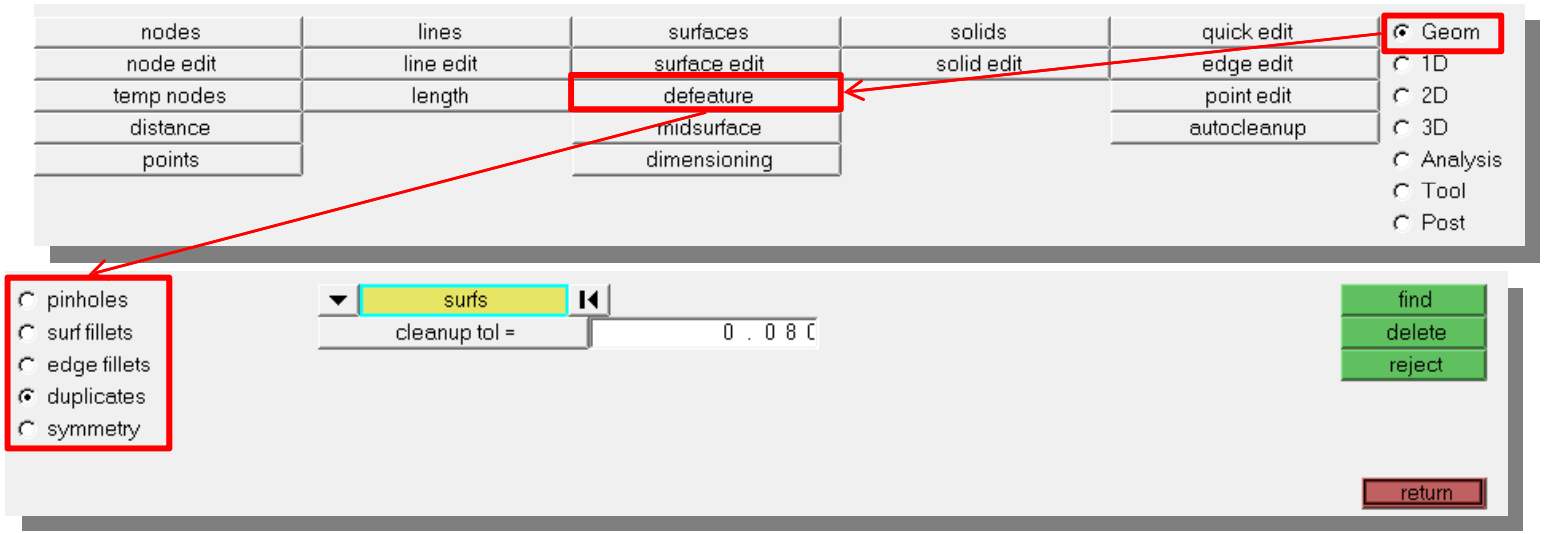
Defeaturing: What is it?

- Depending on the analysis, certain details in the geometry may be ignored. This may depend on:
  - Importance of the part in the overall assembly
  - Location of the feature relative to the area of interest in the analysis
  - Size of the feature vs. the average size of the mesh being used
- Defeaturing is the removal of details in the geometry in order to make the shape of the part simpler



Defeaturing: Tools

- *"Defeature"* panel →*Geom* – defeature



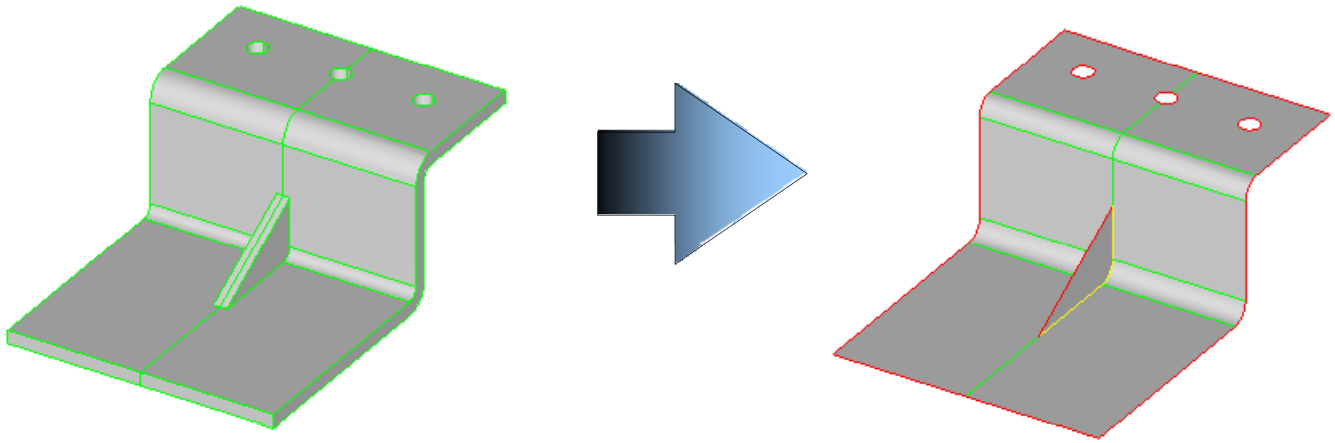
- ***Pinholes:***
  - Searches for closed loops of free edges (holes) within a surface
  - Fills in the holes
  - Leaves fixed point at the center
- ***Surf Fillets:***
  - Searches for surfaces that act as fillet between other surfaces
  - Tangentially extends them to form a sharp corner
- ***Edge fillets:***
  - Searches for rounded corners on a surface
  - Squares off the corner
- ***Duplicates:***
  - Finds and deletes duplicate surfaces
- ***Symmetry:***
  - Finds symmetric surfaces within a single component
  - Deletes or organizes the surfaces into different components

Defeaturing: Tools

- ***Defeature*** panel on ***Geom*** page
  - ***Pinholes:***
    - Searches for closed loops of free edges (holes) within a surface
    - Fills in the holes
    - Leaves fixed point at the center
  - ***Surf Fillets:***
    - Searches for surfaces that act as fillet between other surfaces
    - Tangentially extends them to form a sharp corner
  - ***Edge fillets:***
    - Searches for rounded corners on a surface
    - Squares off the corner
  - ***Duplicates:***
    - Finds and deletes duplicate surfaces
  - ***Symmetry:***
    - Finds symmetric surfaces within a single component
    - Deletes or organizes the surfaces into different components

Midsurfacing: Introduction

- For many FE analyses, parts are represented by shell elements
  - Thickness is assigned mathematically, rather than geometrically
  - Mesh is usually placed on the midplane of the part
- CAD geometry usually comes as a solid part, or a series of surfaces defining a volume.
- Midsurfacing creates a layer of surfaces on the midplane which can be directly meshed

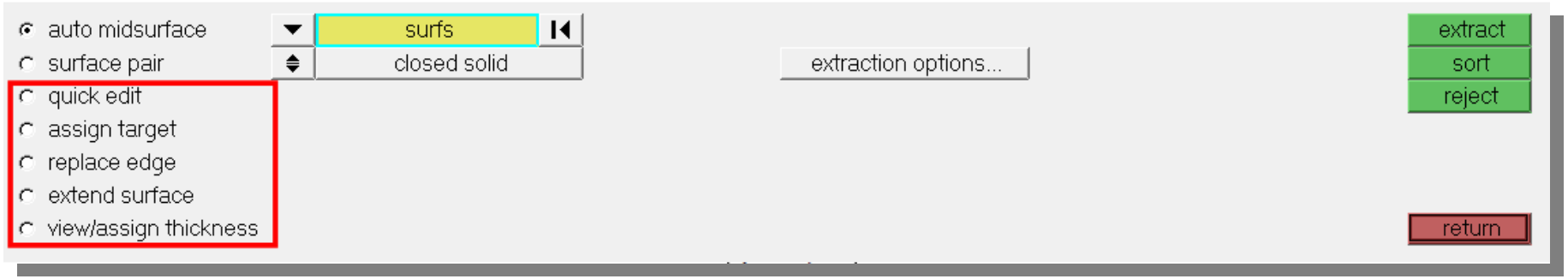


Midsurfacing: Tools

- Midsurfaces can be created using ***midsurface panel*** on the geom page
  - ***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

- Once a midsurface has been created, it can be modified using tools on the ***midsurface panel***
  - ***Quick Edit*** – Repair a midsurface by correcting where the verticies 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: Process & Strategy

1. Obtain a closed volume of surfaces or solids
   - ***Midsurface : auto midsurface*** requires an enclosed volume
   - Use topology repair techniques if needed

2. For complex parts, try defeaturing the surface defining the volume
   - This simplifies the part and may give better results with ***create : solid***

3. Generate the midsurface using ***midsurface : auto midsurface***
   - Use ***surface pair*** for areas that need more control
   - Use ***midsurface : editing tools*** for midsurfaces that need fine tuning

4. View the midsurface and correct errors using the midsurface editing functionalities
   - Can generally use ***quick edit***

Geometry

Midsurface r/t parameter exposed to allow control over the midsurface generation in areas with high radius-to-thickness ratios

9.0 - r/t parameter was hard-coded to 2.0

10.0 - r/t parameter is user-defined with default of 2.0