

Module 03: Repairing Geometry

Release 2022 R1

Please note:

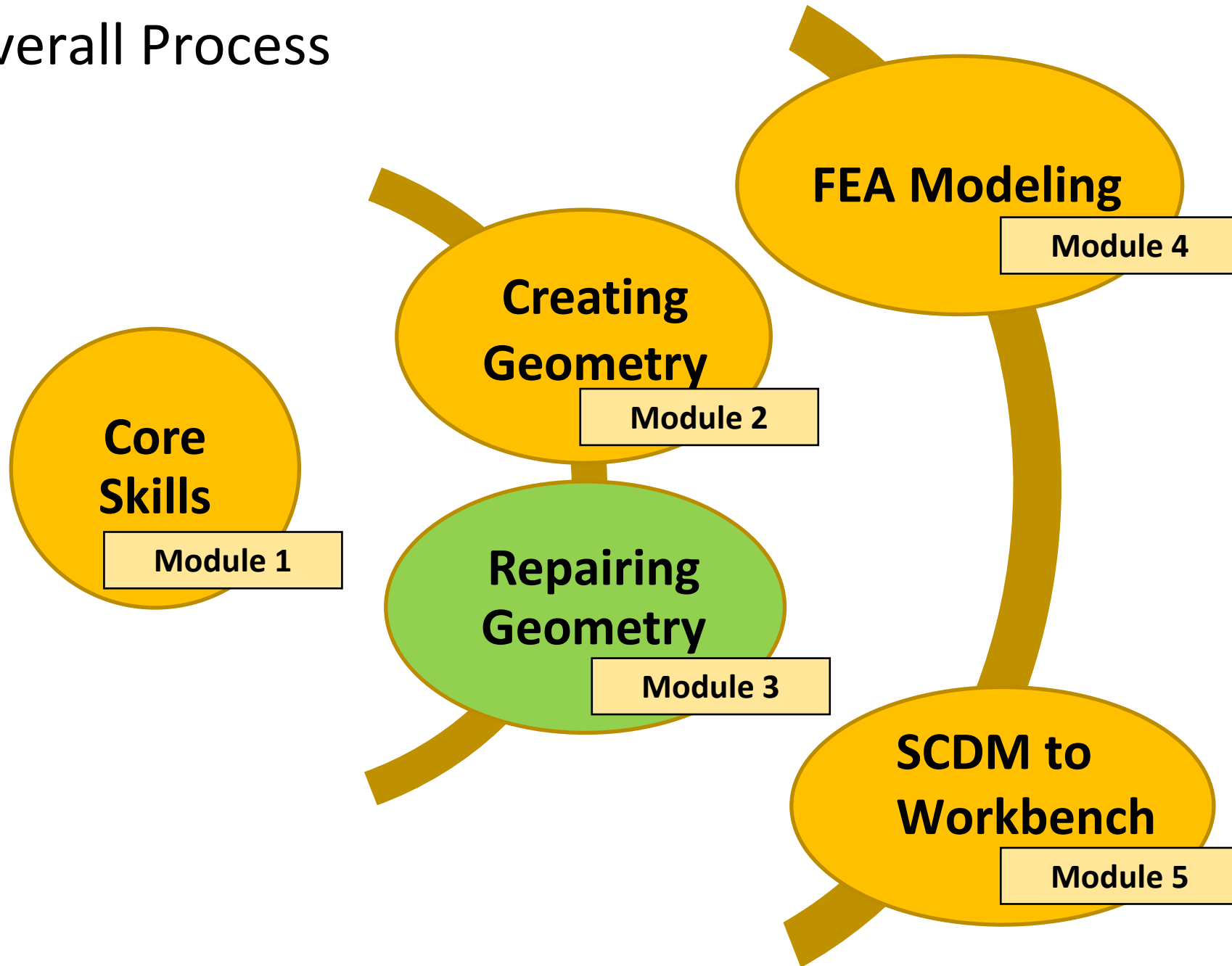
- These training materials were developed and tested in Ansys Release 2022 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.



Learning Outcomes

- Importing Geometry in Ansys SpaceClaim
- Why we need to repair an imported geometry
- What are the common issues found in an imported geometry
- Fix Issues One-by-One Vs All-at-Once
- Discover some specialized tools for repair
- Introduction to how we can handle STL Geometry

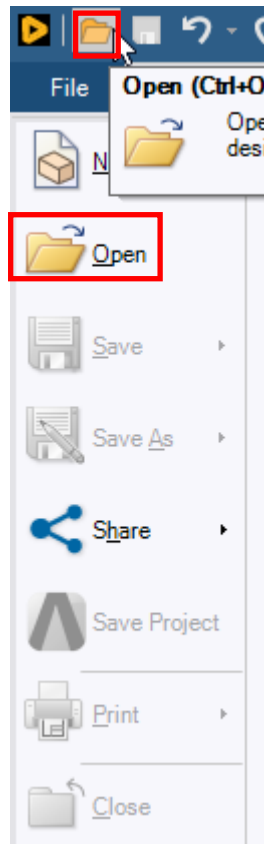
Overall Process



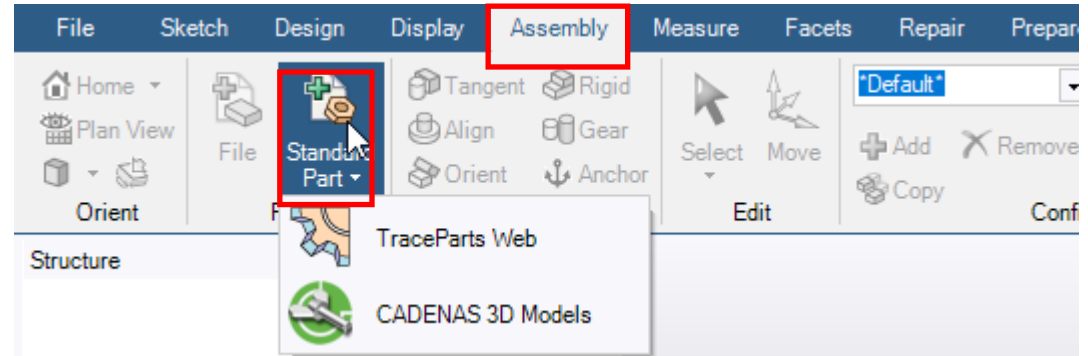
Importing Geometry (1)

- Two Import Options

Open CAD file in new session



Insert CAD file to existing session from 2 different sources



Importing Geometry (2)

• Supported Files

- Supports import from major CAD packages (CATIA V4, Pro/E, NX, Solid Works, etc.)
 - Separate license not required
- Neutral file formats like STEP and Parasolid are also supported
- Additional license is required for
 - Faceted Data Toolkit
 - JT Exchange
 - 3D PDF
 - CATIA V5 (special interface for SpaceClaim)
 - CATIA V6 (same interface as Workbench)
- Bi-directional Attach mode with other CAD software is supported via the “Blocks’ feature

File Formats

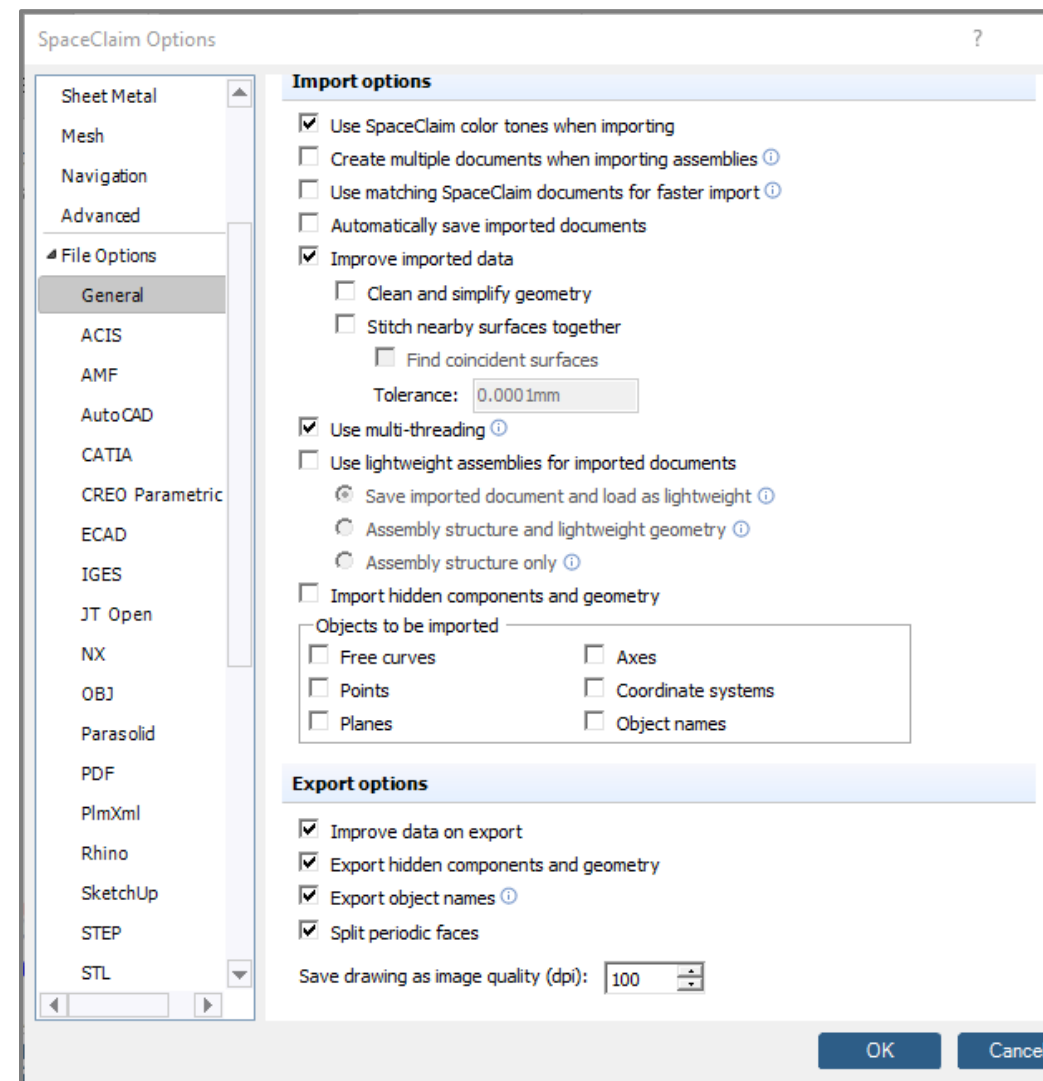
SpaceClaim files (*.scdoc)
All Supported Files (*.*)
Discovery files (*.dsco)
ACIS (*.sat;*.sab;*.asat;*.asab)
AMF (*.amf)
Ansys (*.agdb;*.pmdb;*.meshdat;*.mechdat;*.dsdb;*.cmdb;*.dbs)
Ansys Electronics Database (*.def)
AutoCAD (*.dwg;*.dxf)
CATIA V4 (*.model;*.exp)
CATIA V5 (*.CATPart;*.CATProduct;*.cgr)
CATIA V6 (*.3dxml)
CREO Parametric (*.prt;*.xpr;*.asm;*.xas*)
DesignSpark (*.rsdoc)
ECAD (*.idf;*.idb;*.emn)
Fluent Mesh (*.tgf;*.msh)
ICEM CFD (*.tin)
IGES (*.igs;*.iges)
Inventor (*.ipt;*.iam)
JT Open (*.jt)
NX (*.prt)
OBJ (*.obj)
OpenVDB (*.vdb)
OSDM (*.pkg;*.bdl;*.ses;*.sda;*.sdp;*.sdac;*.sdpc)
Other ECAD (*.anf;*.tgz;*.xml;*.cvg;*.gds;*.sf;*.strm)
Parasolid (*.x_t;*.xmt_txt;*.x_b;*.xmt_bin)
PDF (*.pdf)
PLM XML (*.plmxml;*.xml)
PLY (*.ply)
QIF (*.QIF)
Revit (*.rvt;*.rfa)
Rhino (*.3dm)
SketchUp (*.skp)
Solid Edge (*.par;*.psm;*.asm)
SolidWorks (*.sldprt;*.sldasm)
SpaceClaim Script (*.scscript;*.py)
SpaceClaim Template (*.scdot)
STEP (*.stp;*.step)
STL (*.stl)
VDA (*.vda)
VRML (*.wrl)
All Files (*.*)

Importing Geometry (3)

- Import Options

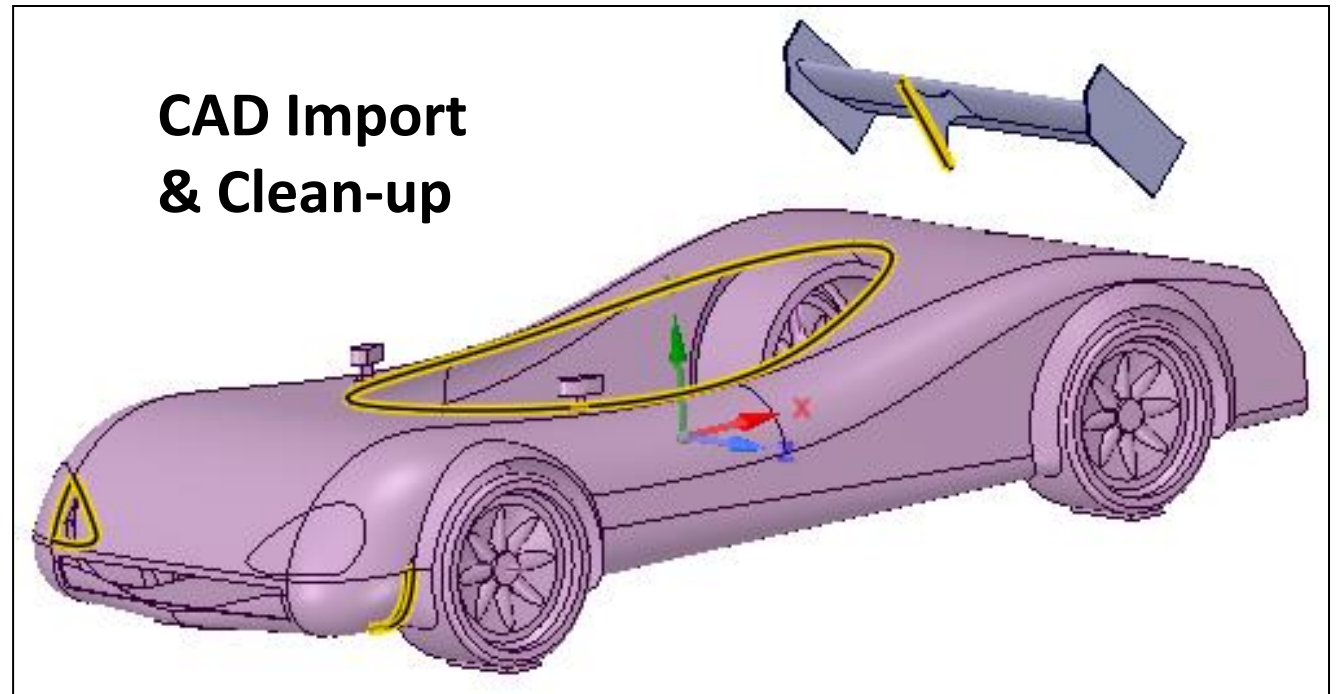
- There are advanced options available under SpaceClaim Options tab in File Menu
 - Useful for optimizing the importing and exporting process for your needs
- Several translation methods available to enable data exchange with CAD/CAE systems
 - Direct Integration/CAD Readers
 - Import of generic CAD formats (IGES, ACIS etc)
- Parameters, Named Selections, and Attributes can not be passed from CAD to SpaceClaim as SpaceClaim uses reader mode for all CAD files.
- Enclosure and Symmetry Processing is not available
- Work points (construction points) are not transferred

SpaceClaim Options



Why the Need for Repair?

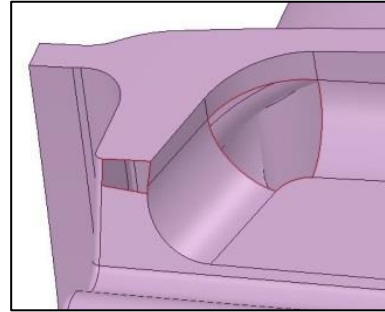
- Translation can:
 - Return incomplete, corrupt, or disconnected geometry
 - Requires repair
 - Return geometry details unnecessary for CAE analysis
 - Requires defeaturing
- These issues must be fixed to
 - Create watertight fluid bodies
 - Prevent meshing issues



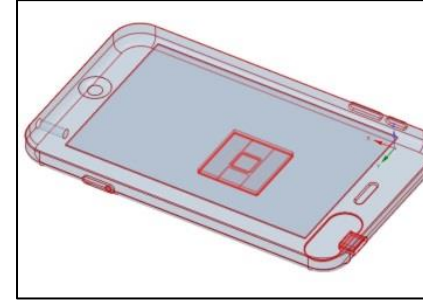
Common Issues Found in Geometry (1)

- Many potential issues
 - Missing faces
 - Small faces
 - Free faces
 - Sharp angles
 - Over detailed features
 - Hard edges
 - Small edges
 - Others ...
- These problems can be fixed using several tools available under "Repair" menu

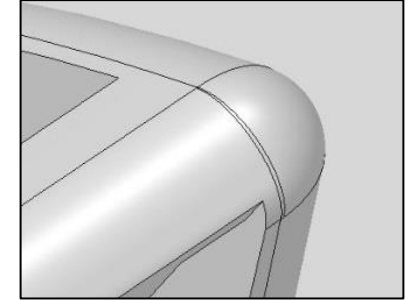
Missing Faces



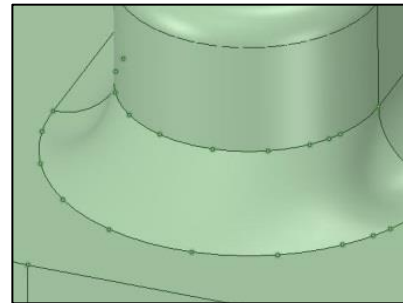
Free Faces



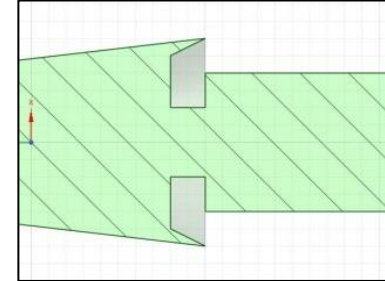
Small Faces



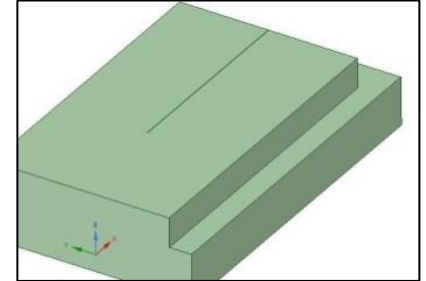
Small Edges



Sharp Angles



Hard Edges







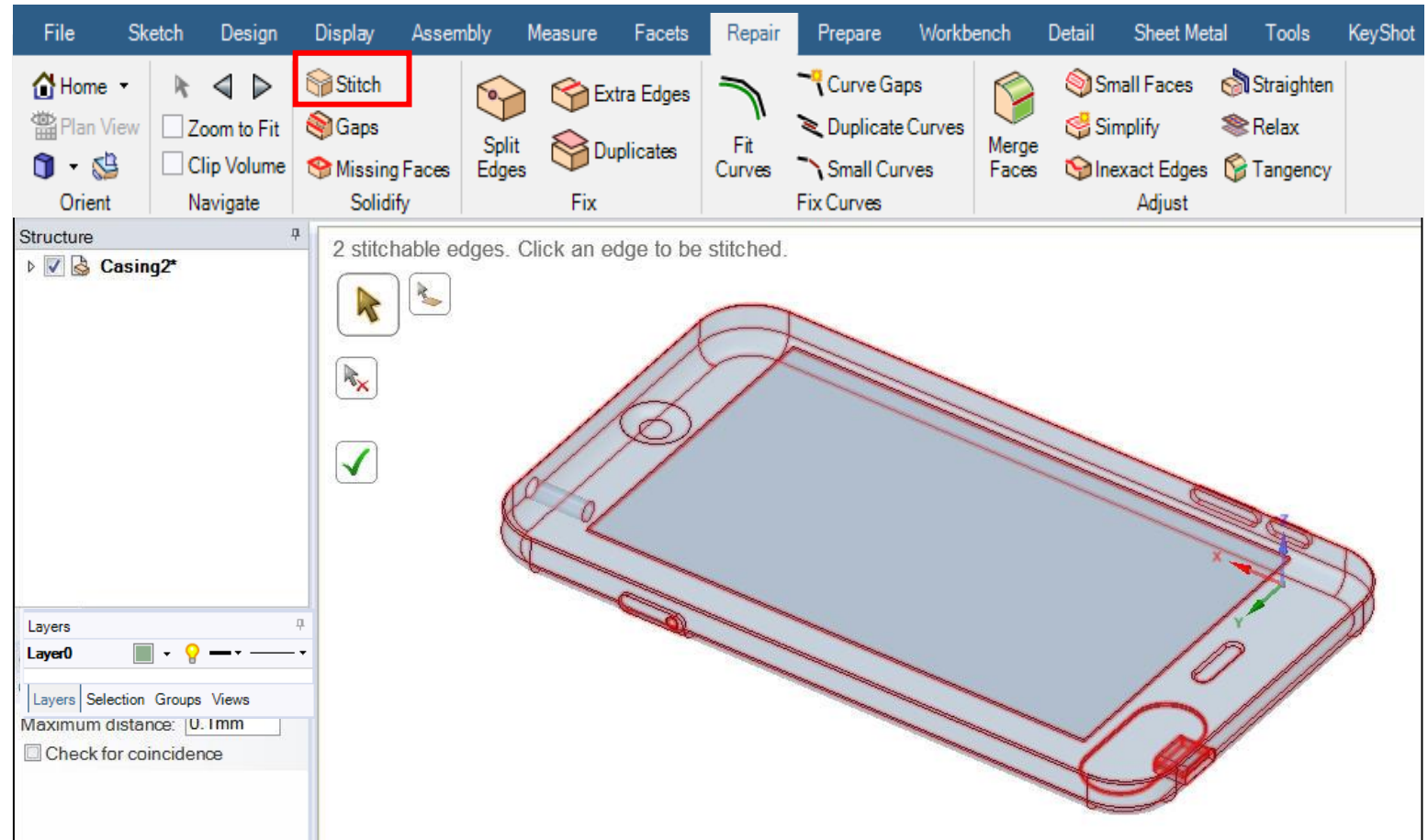
Common Issues Found in Geometry (2)

- Stitch

- The Stitch tool combines surface part faces that are touching at their edges

- Tool Guides Options

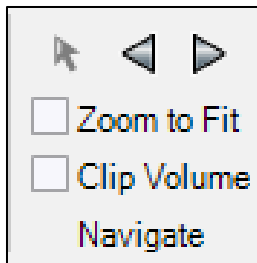
-  Selects the location to repair one-by-one
-  Select and add locations to auto selection
-  Removes the location from selection
-  Repair all locations at a time



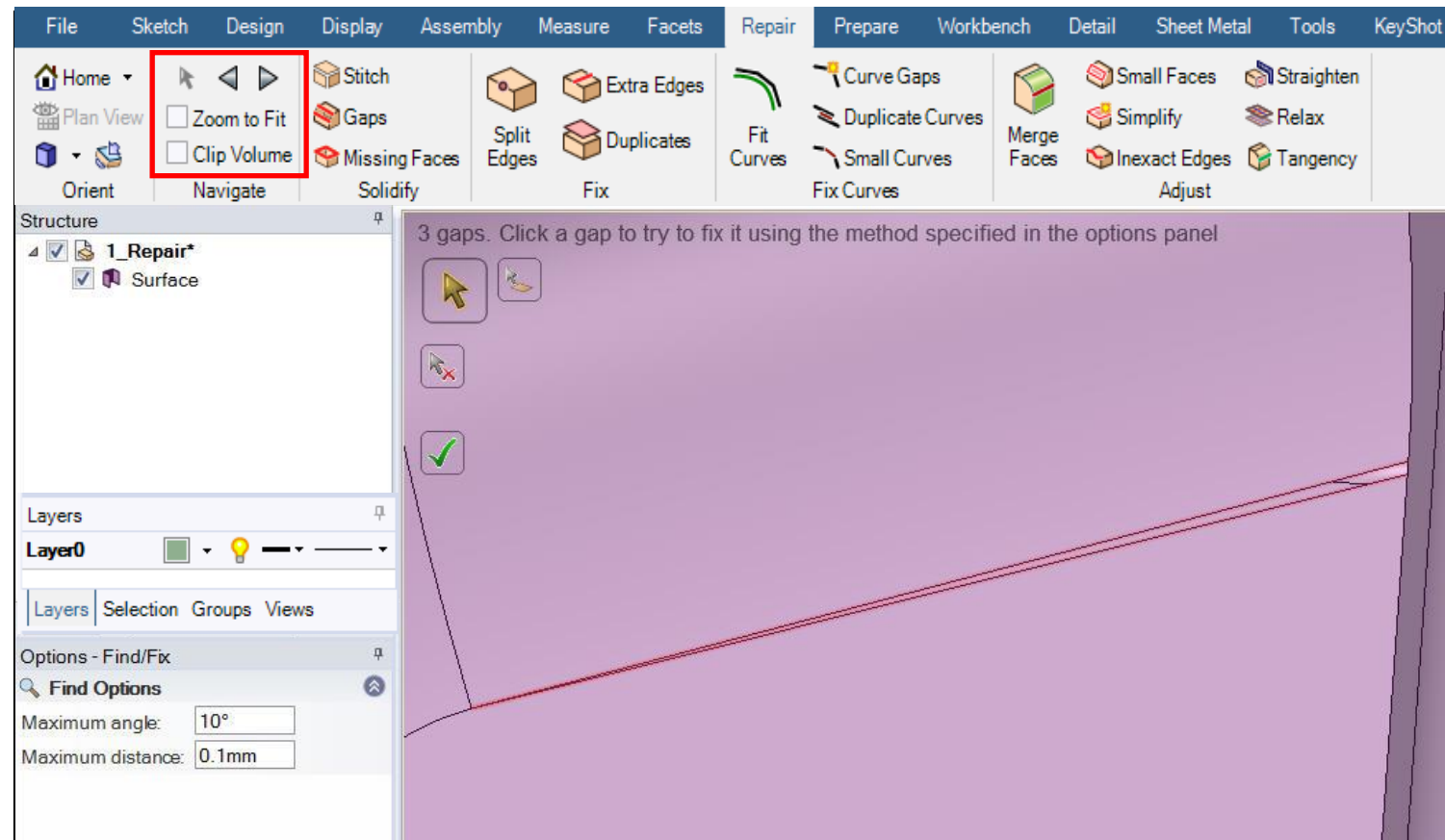
Common Issues Found in Geometry (3)

- Gaps

- The Gaps tool removes gaps between faces
- Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it

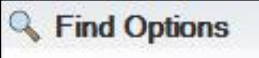
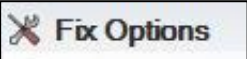


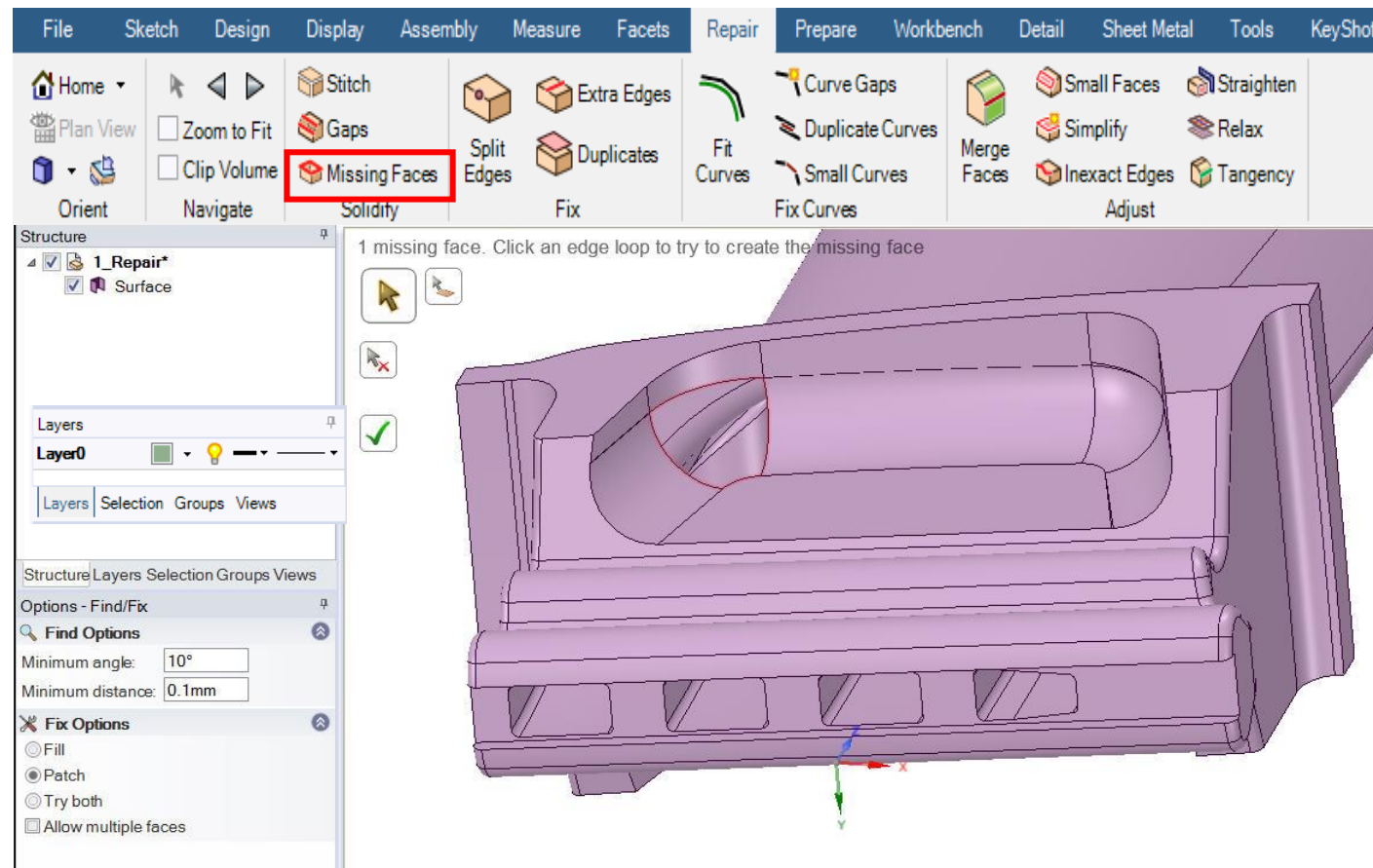
- Click “Next” or “Previous” to step through and highlight each identified problem
- Select “Zoom to Fit” if you want to automatically zoom in on the problem



Common Issues Found in Geometry (4)

- Missing Faces

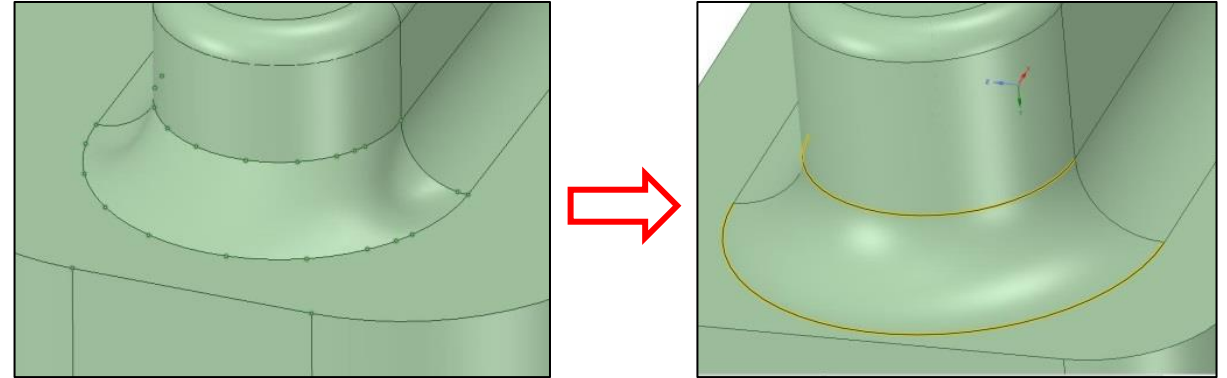
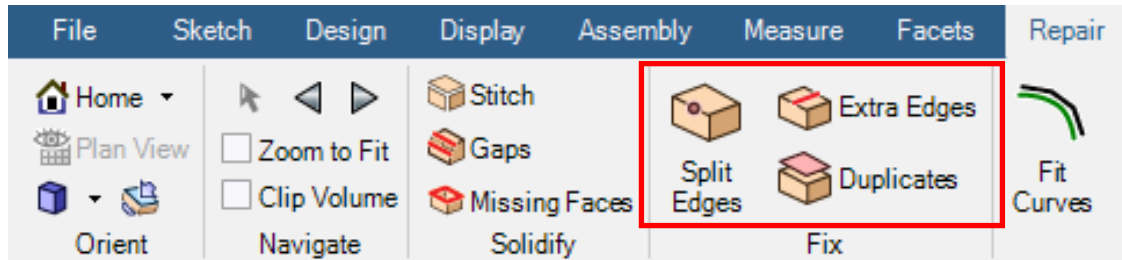
- This tool automatically detects and fills missing faces of an object
- This can fill the missing faces all-by-one or one-by-one
- Find Options 
 - Set criteria for auto problem selections
- Fix Options 
- Different approaches to fill the missing faces



Common Issues Found in Geometry (5)

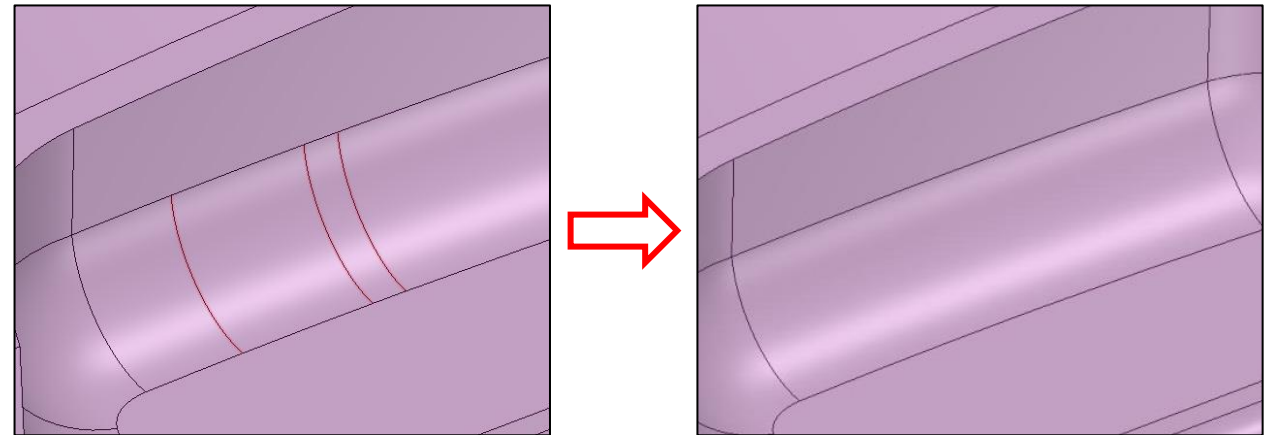
- Split Edges

- The Split Edges tool detects and merges coincident edges that do not mark the boundaries of new faces



- Extra Edges

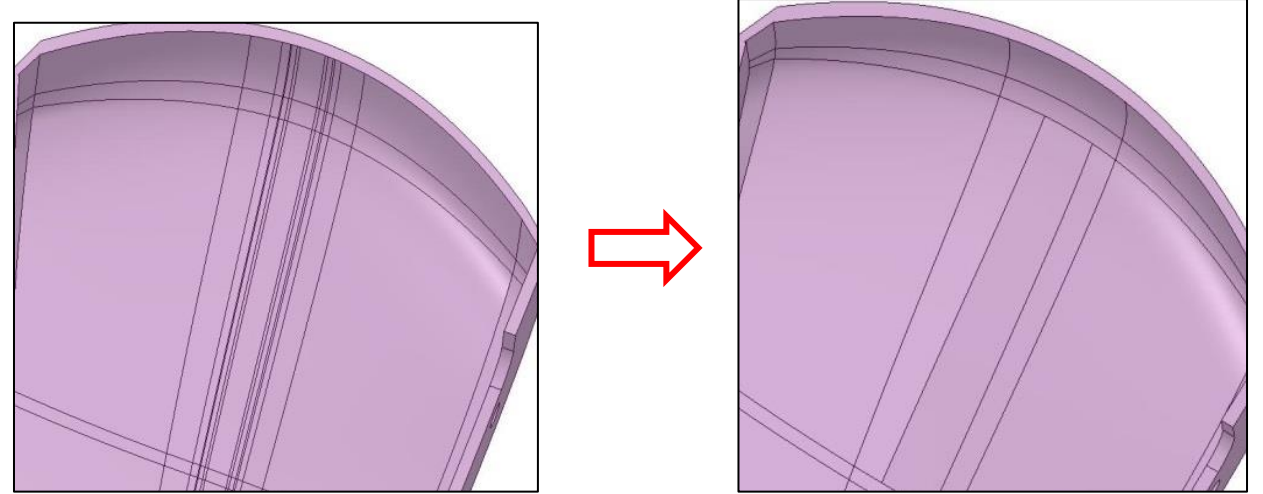
- The Extra Edges tool works like Merge Faces but removes the edges between the faces
- Merging faces simplifies the model by removing edges and makes the model more difficult to modify



/ Common Issues Found in Geometry (6)

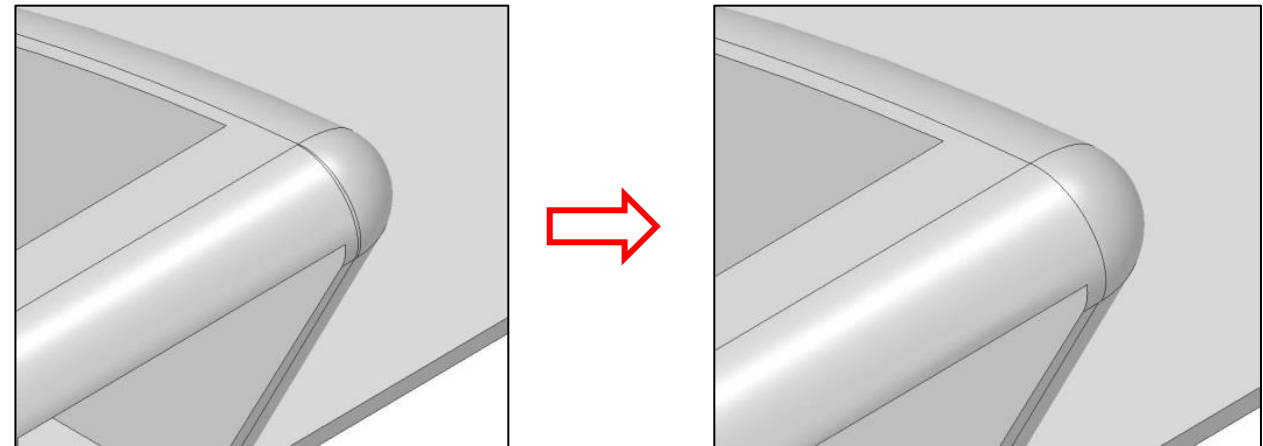
- Merge Faces

- The Merge Faces tool replaces two or more neighboring faces with a single new face that closely fits the original faces



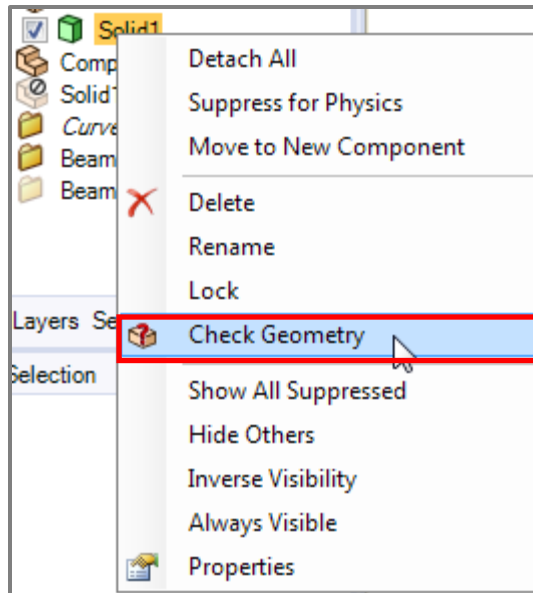
- Small Faces

- The Small Faces tool detects and removes small and sliver faces

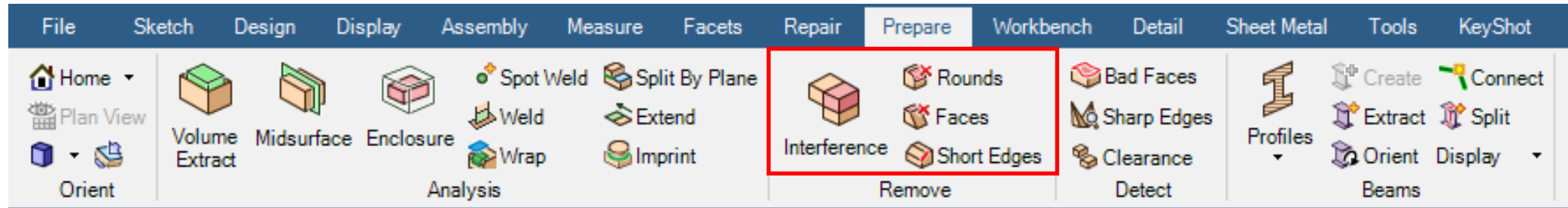


/ Fix Issues One-by-One Vs All-at-Once

- SCDM allows fixing issues found at all locations at the same time
- This is useful when repairing large models
- But while fixing it may create invalid patches (surfaces) in the geometry
- It is advisable to double-check the locations fixed by automatic operations
- Check Geometry tool can be effectively used to validate the fixes

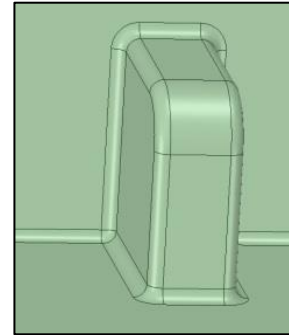


Specialized Tools (1)

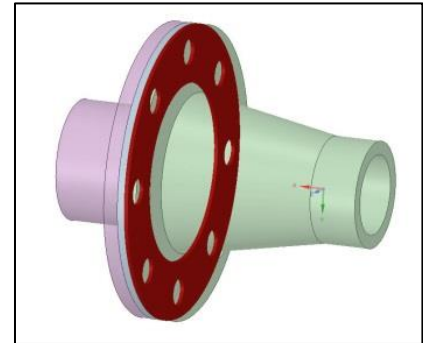


- Actual designs created have lot of features which are unnecessary for CAE analysis
- These features can be simplified using several Remove tools available under Prepare tab
- Specialized tools are used to remove features
 - Rounds
 - Interferences
 - Faces
 - Short Edges

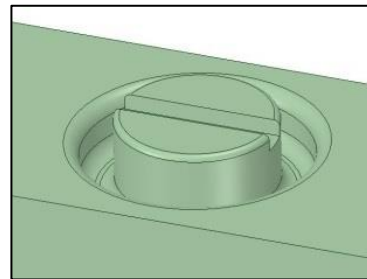
Rounds



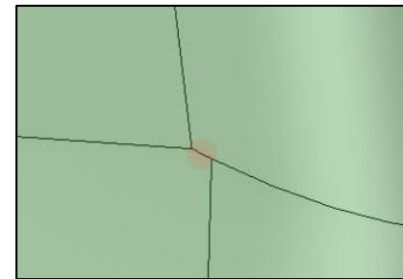
Interference



Faces



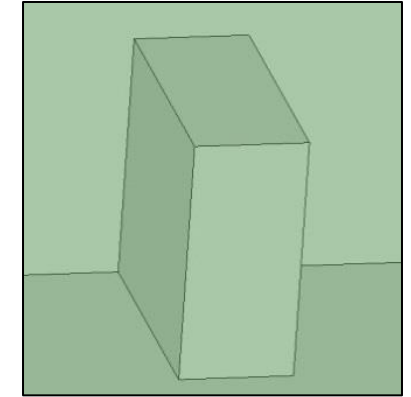
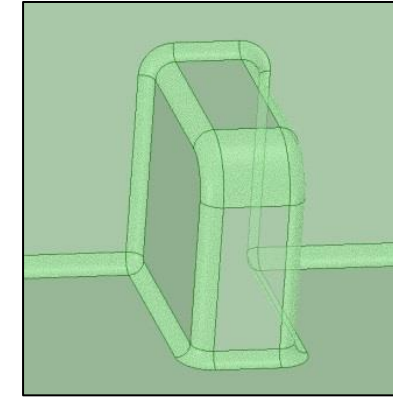
Short Edges



/ Specialized Tools (2)

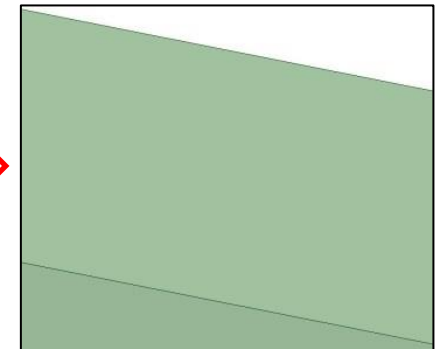
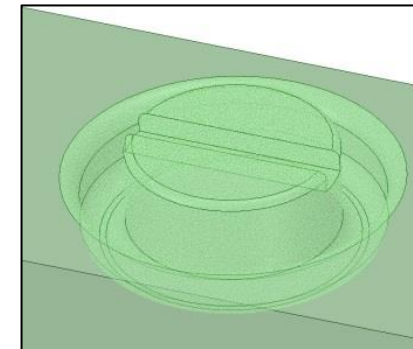
- Rounds

- It is similar to the Fill tool, except it only selects rounded edges
- You will have a greater chance of success if you remove only a few rounds at a time



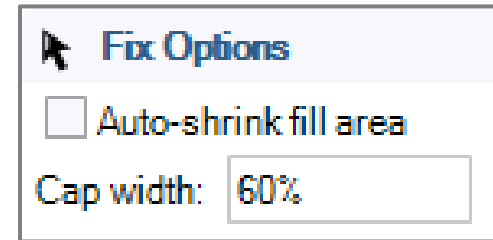
- Faces

- The Faces tool allows you to quickly remove faces from your design. Use it to simplify your design by removing holes, protrusions, etc.

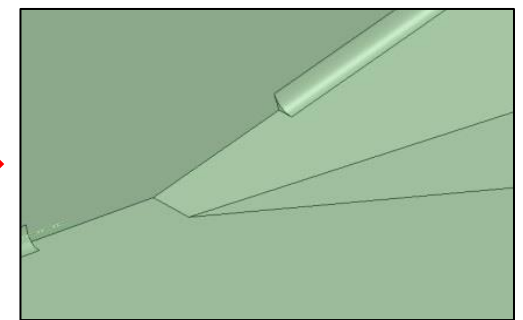
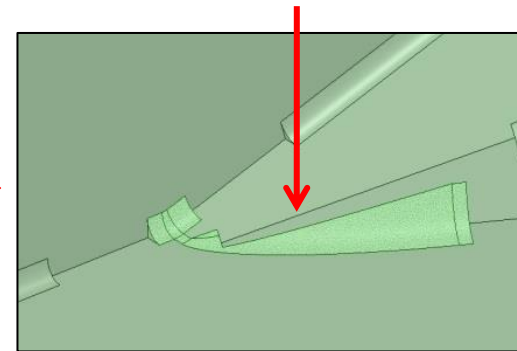
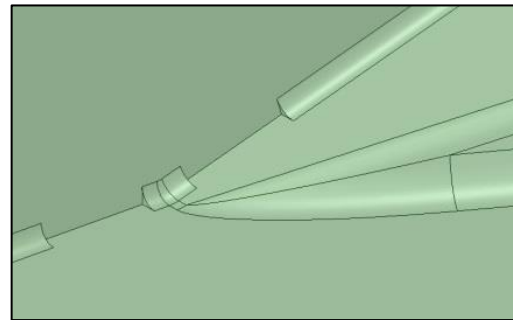
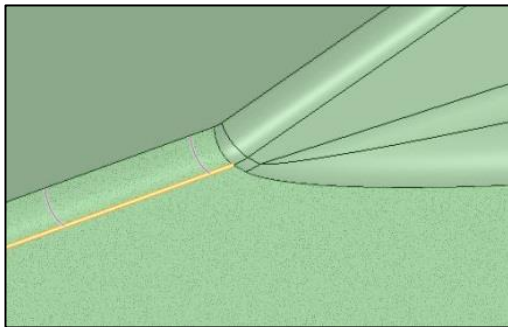


Specialized Tools (3)

- Split and partially remove a round:
 - When a chain of rounds cannot be filled in its entirety, the rounds will be partially removed
 - Click on the edge to split the rounded face
 - From the Fix Options panel, change the Cap width option to adjust the percentage of the round face that is removed



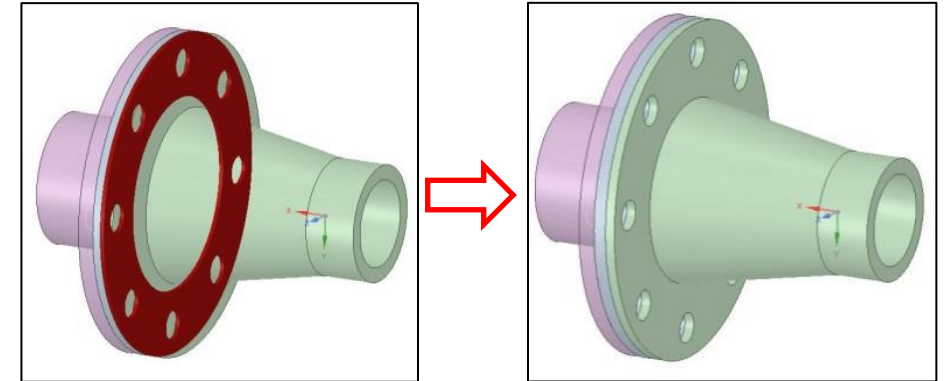
Use “Faces” tool to remove remaining part of fillets



Specialized Tools (4)

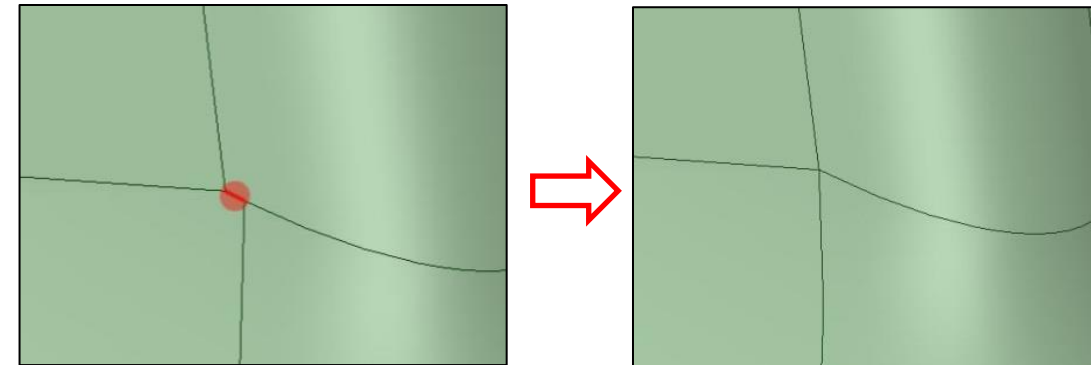
- Interference

- The tool searches all visible bodies for interference
- The interference is removed from the body with the largest volume



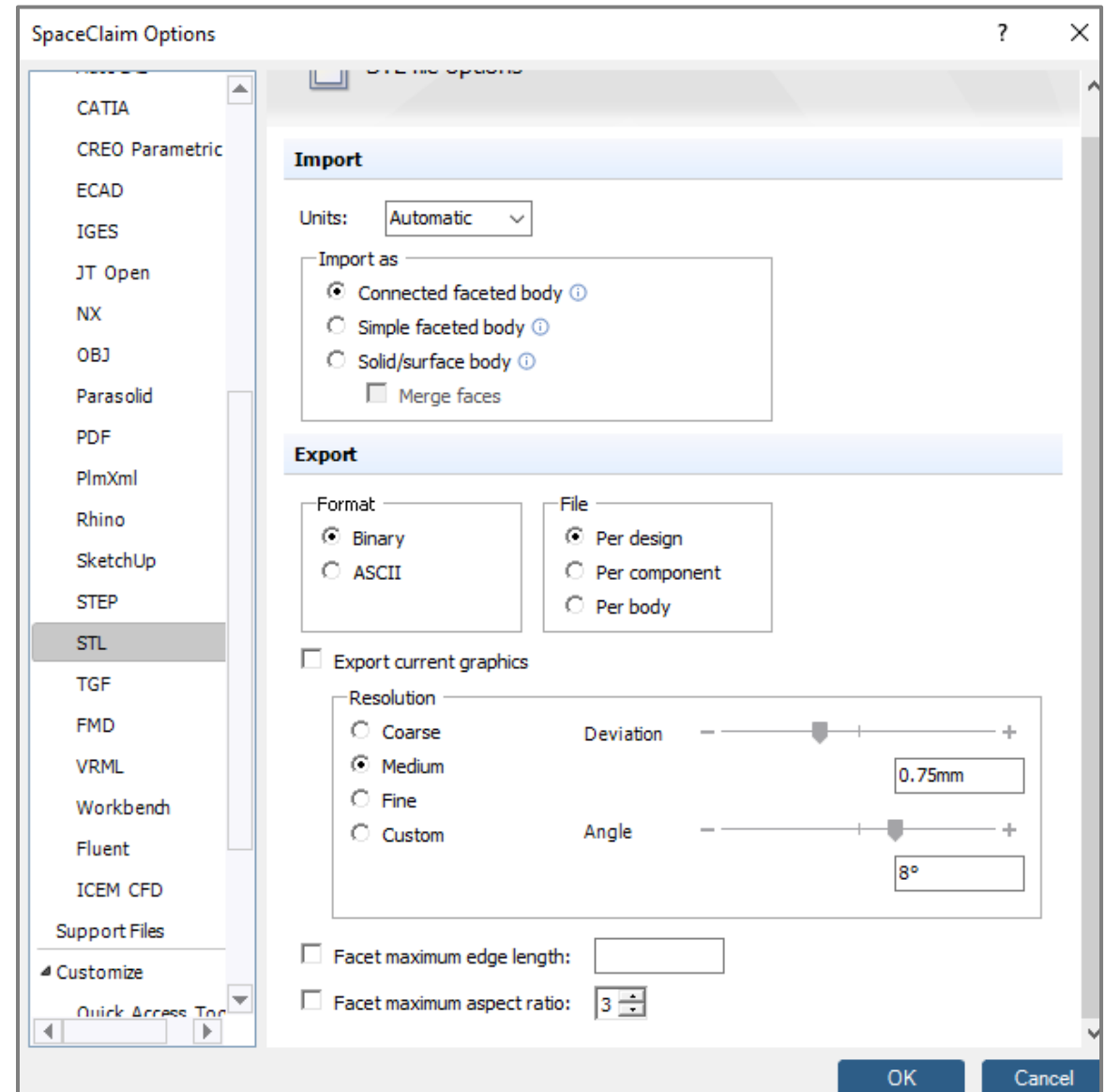
- Short Edges

- This removes the small edges coincident with boundary edges
- It connects the two vertices of the short edge to eliminate it



Convert STL Mesh to Geometry (1)

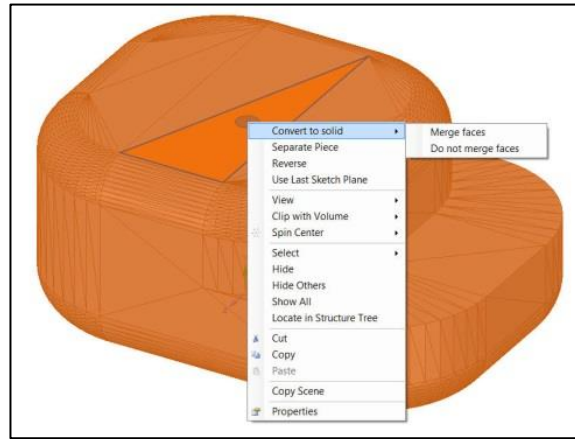
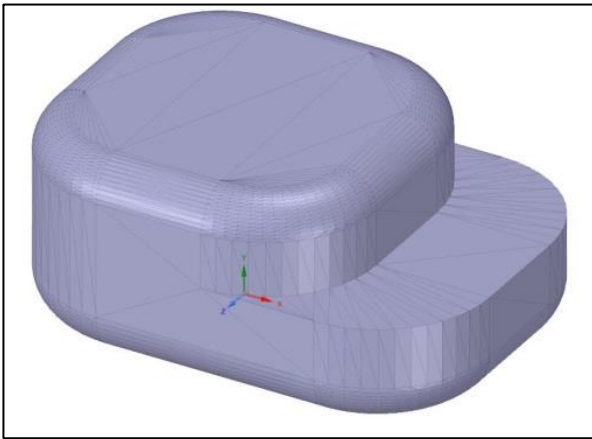
- STL Import Options
 - Similar to other CAD formats, STL import can be optimized through SpaceClaim Options tab
- Import as:
 - Connected mesh: Create a lightweight mesh with full connectivity information between facets
 - Simple mesh: Create a lightweight mesh with no connectivity information between facets
 - Solid/surface body: Convert STL triangles into full geometry
 - Check the Merge faces checkbox to seamlessly merge one or more faces



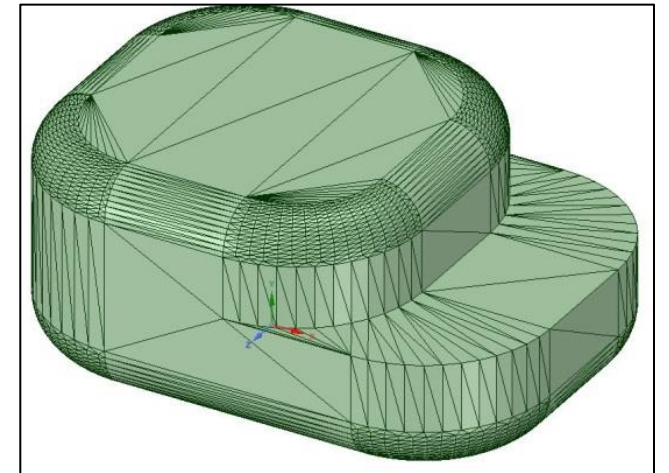
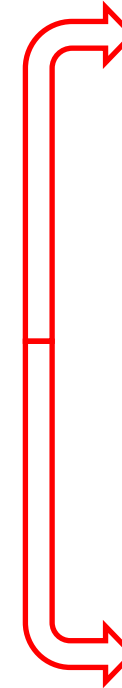
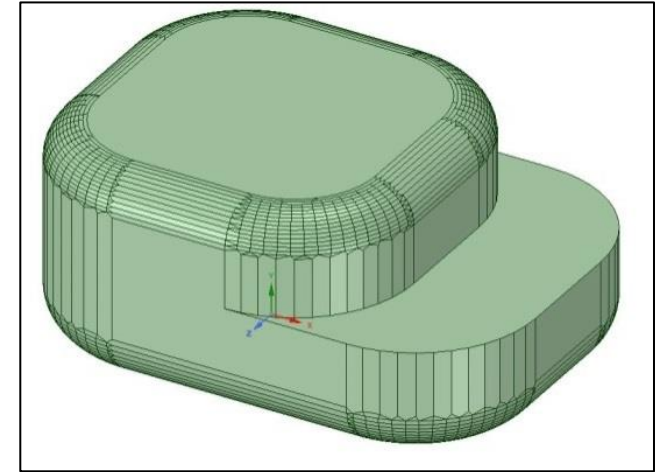
Convert STL Mesh to Geometry (2)

- Converting to CAD

- You can convert mesh files into CAD
- Options are available if you right click on object in Graphics or Structure Tree
- Options are available to convert by merging the faces or without merging them

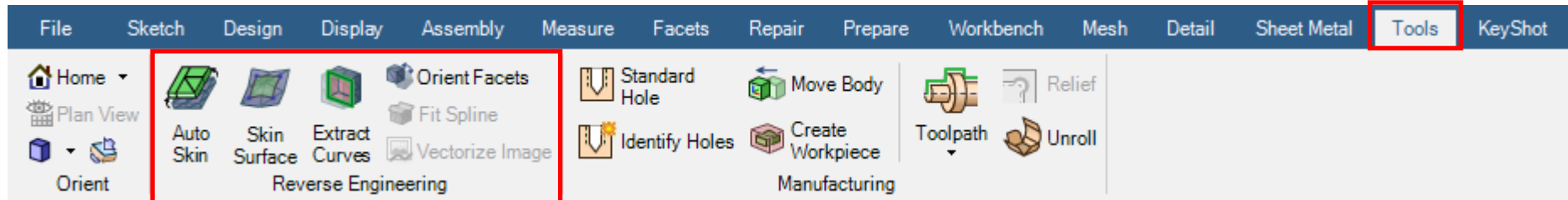


Merge Faces

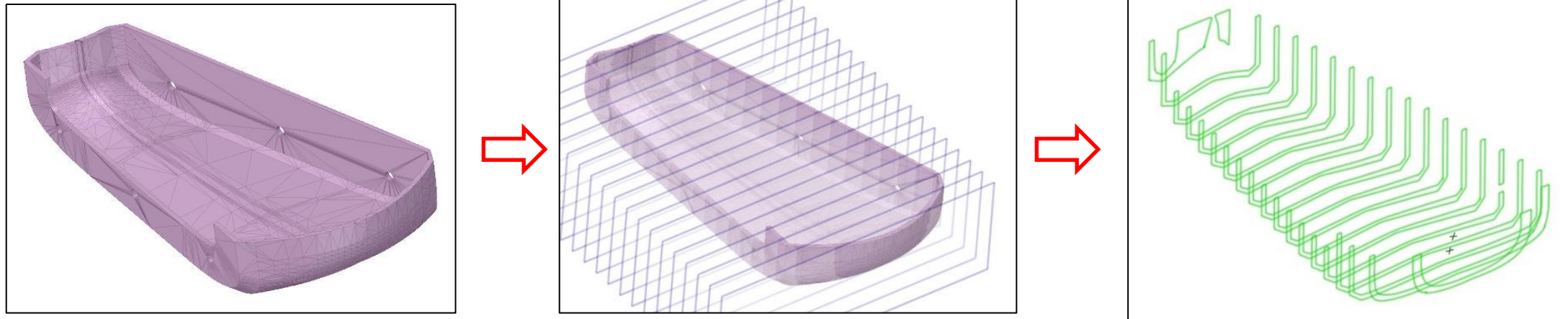


Do Not Merge Faces

Convert STL Mesh to Geometry (3)



- Extract Curves
 - Creates curves at the intersection of object cross-sections and planes



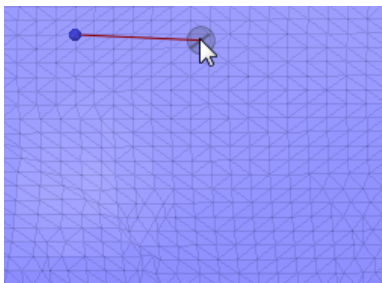
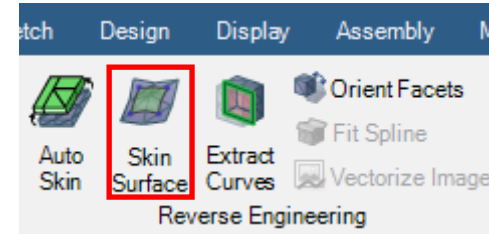
Convert STL Mesh to Geometry (4)

- Skin Surfaces

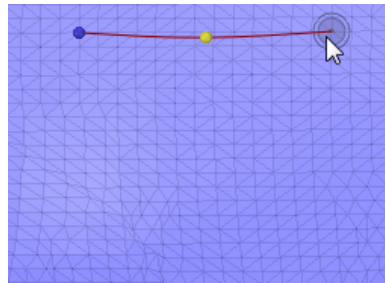
- Helps to reverse-engineer a surface model

From faceted data

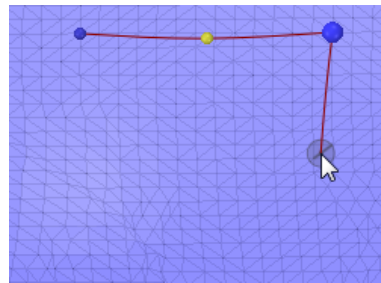
1. Single-click on the mesh where you want to start the sketch
2. Single-click to create internal edge points
3. Double-click to end the edge and create a corner
4. Continue creating edges and corners and close the shape by single-clicking on the start point
5. Hit the escape key to remove the current sketch. Click Complete to create the surface



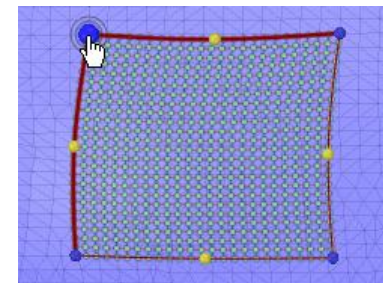
1



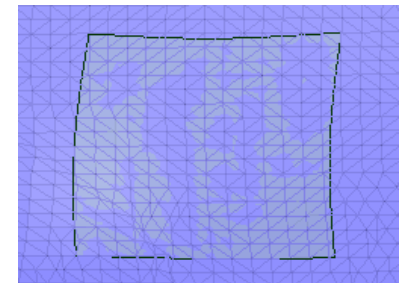
2



3



4

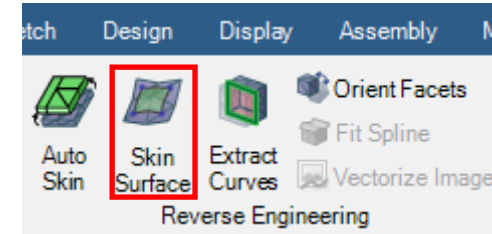


5

Note: You can edit a sketch by dragging a corner or by adding or dragging a control point

/ Convert STL Mesh to Geometry (5)



- Skin Surfaces Options

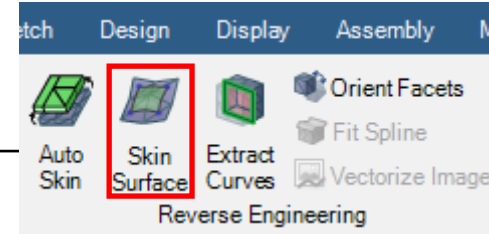


Samples	The value entered determines the number of Control Points on the Control Curves for the surface. By default, the number of sample points is fixed and cannot be changed. If you check the Full Preview option, you can enter a new number of sample points. In Full Preview mode you can enter a new number of points directly, or use the dropdown slider.
Full Preview	When you check Full Preview ON, you can control the number of sample points used in the surface. In Full Preview mode, when you change the number of sample points, the preview dynamically updates. The speed of the dynamic updates is affected by the number of sample points. Fewer points leads to faster updates.

/ Convert STL Mesh to Geometry (6)

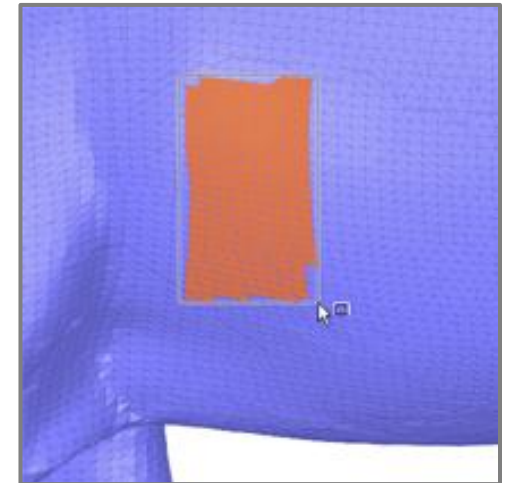
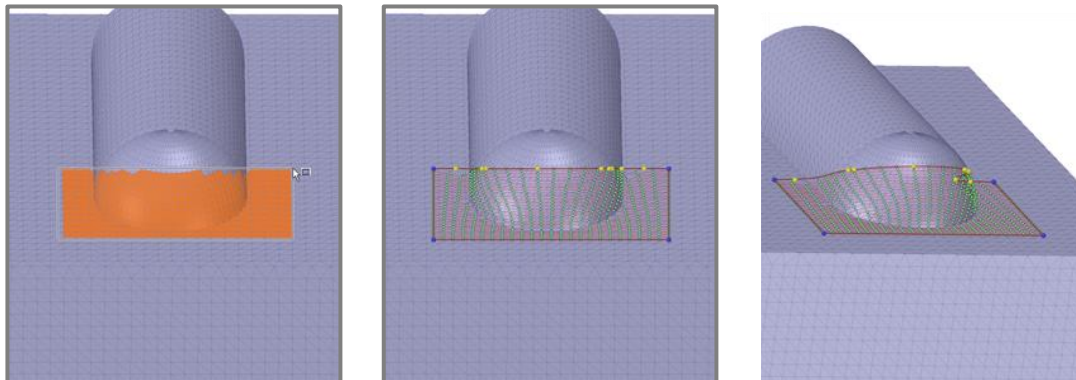
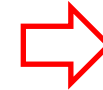
- Skin Surfaces Tool Guides

	The Select Boundary sketch surfaces patch boundaries on the facets
	The Select Geometry tool guide helps to choose planes or existing patch edges to create boundary loops for surface patches



- Using box-select

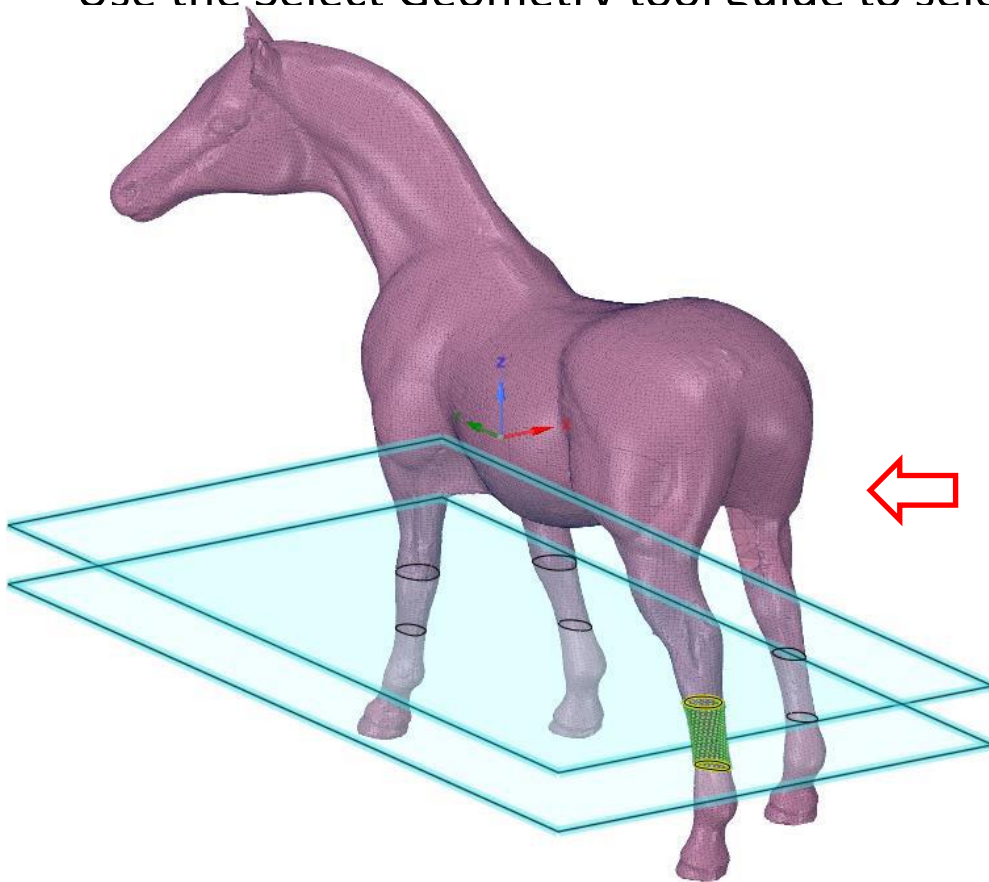
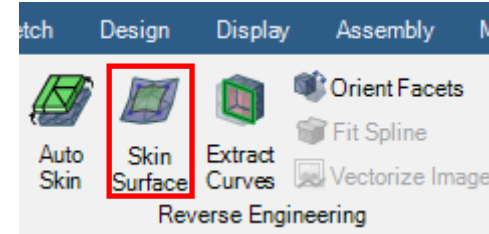
- We can simply box-select the area where we want the sketch
- Control points are automatically added in areas with curvature



Convert STL Mesh to Geometry (7)

- Periodic Surfaces

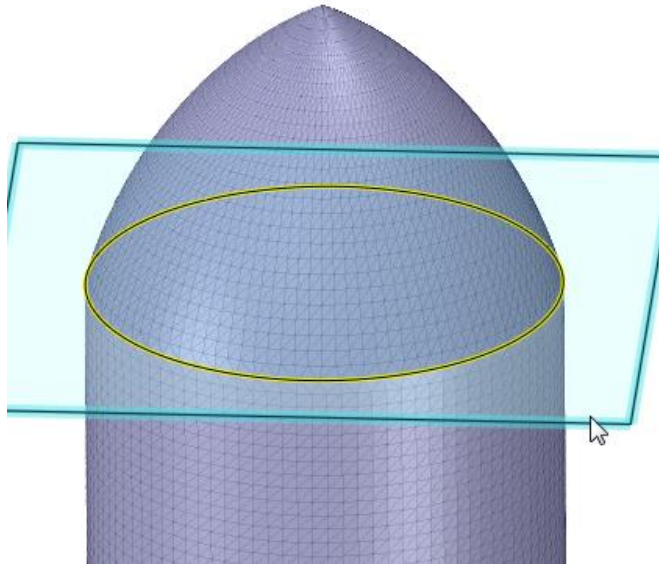
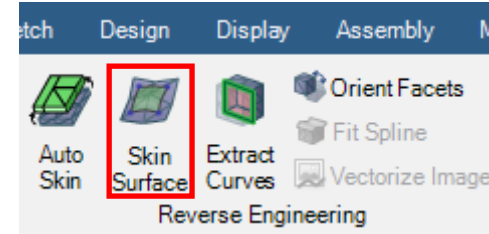
- Use the Select Geometry tool guide to select planes and create sections



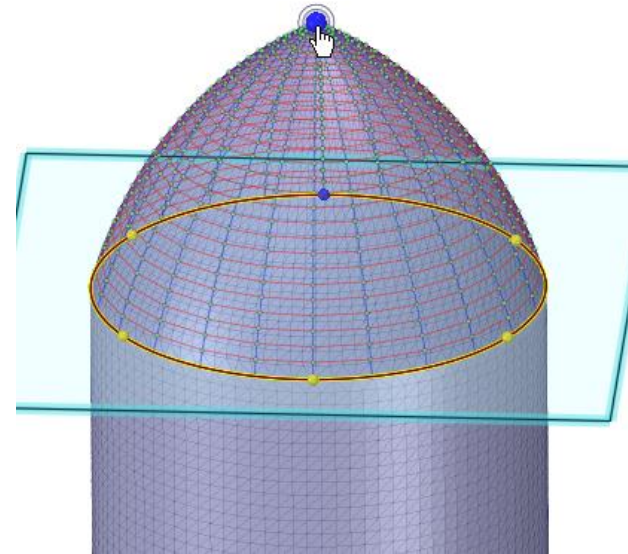
The tool displays preview for the closest loop. If there are multiple loops created. You can explicitly select a different loop if the default that is chosen is not the one you want.

/ Convert STL Mesh to Geometry (8)

- Periodic Surfaces
 - Periodic surfaces can also be conical



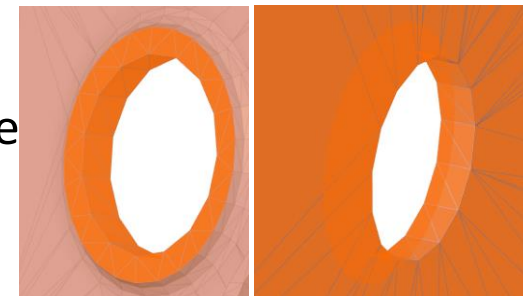
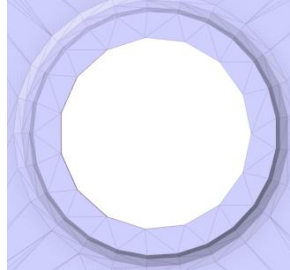
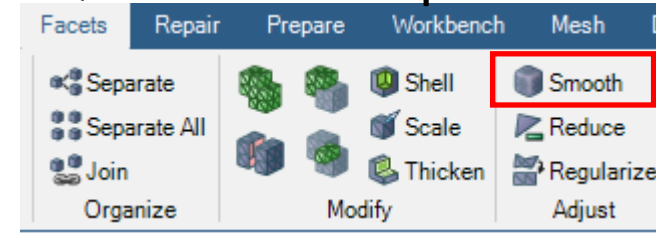
Create a periodic edge on the section



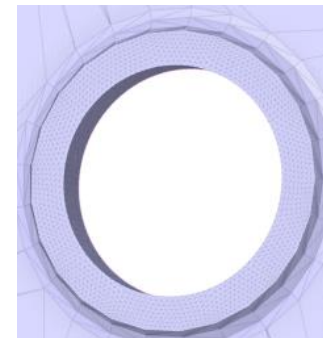
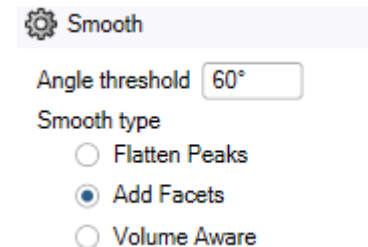
Click on the apex of the cone to preview the surface

Quick Workshop 03.1

- Open a new SpaceClaim project and import the stl file QuickWorkshop03.1
- Go to the Facets tab and click on the Smooth button
- The aim is to add more triangles to hole
- Double click on the surfaces at the top pf the hole
 - Hold the control key down, double click on the surfaces inside the hole
 - Hold the control key down and double click on the surfaces at the bottom of the hole



- In the details of 'Smooth' choose Add Facets
- Click three times on the green tick mark
- Check the triangles distribution around the hole

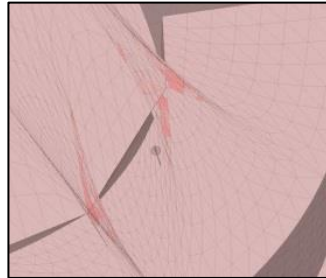


/ Handling STL Geometry (1)

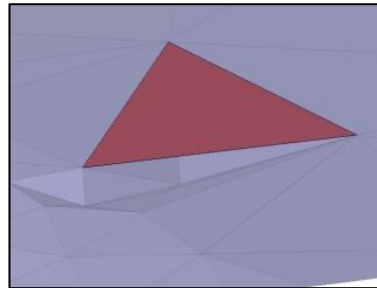
- Issues with STL data

- Intersections
- Non-manifold/ Over connected
- Holes/Gaps
- Sharp Features

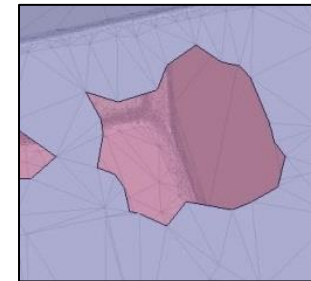
Intersection



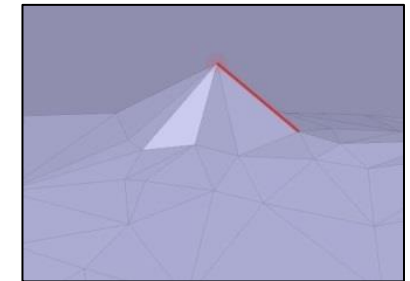
Non-manifold



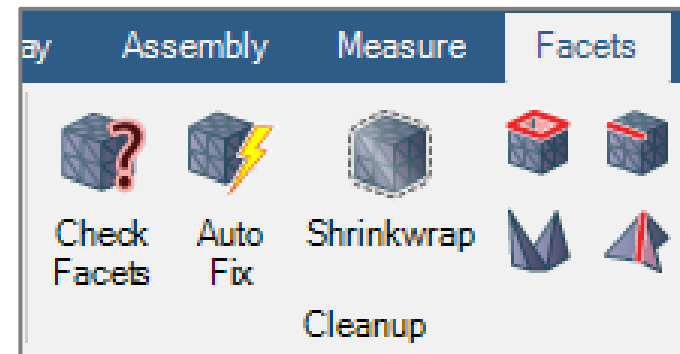
Holes



Sharps



- There are various tools available under Facets' menu to fix the issues related to STL data

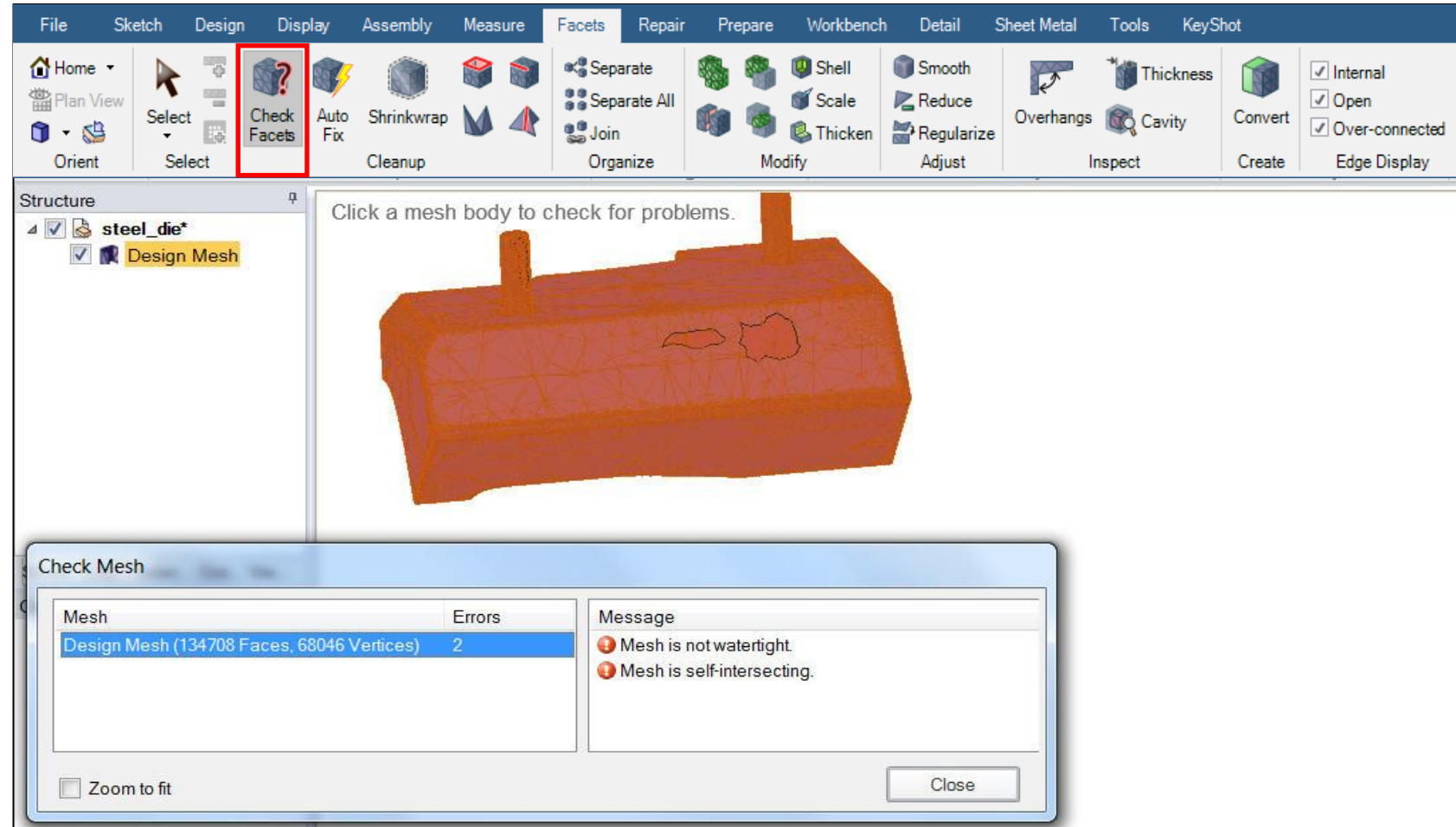


Handling STL Geometry (2)

- Check Facets

- Use the Check Facets tool to list problems with the Mesh such as

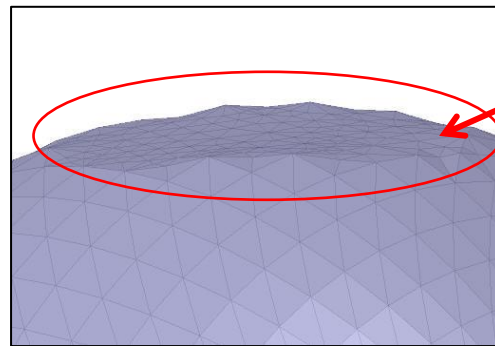
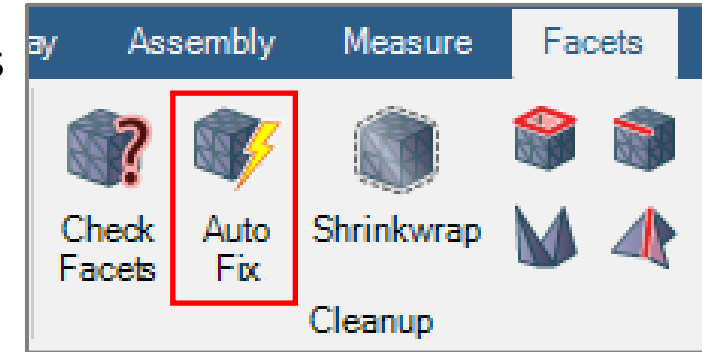
- Intersection
- Non-manifold connection
- Multiple faces
- Holes



/ Handling STL Geometry (3)

- Auto Fix

- Use the Auto Fix tool to automatically search for, and fix, any mesh defects
- The Auto Fix tool performs the following sequence:
 1. Fix self-intersections
 2. Close holes
 3. Recheck for self-intersections
 4. Fix self-intersections
 5. Remove void shells

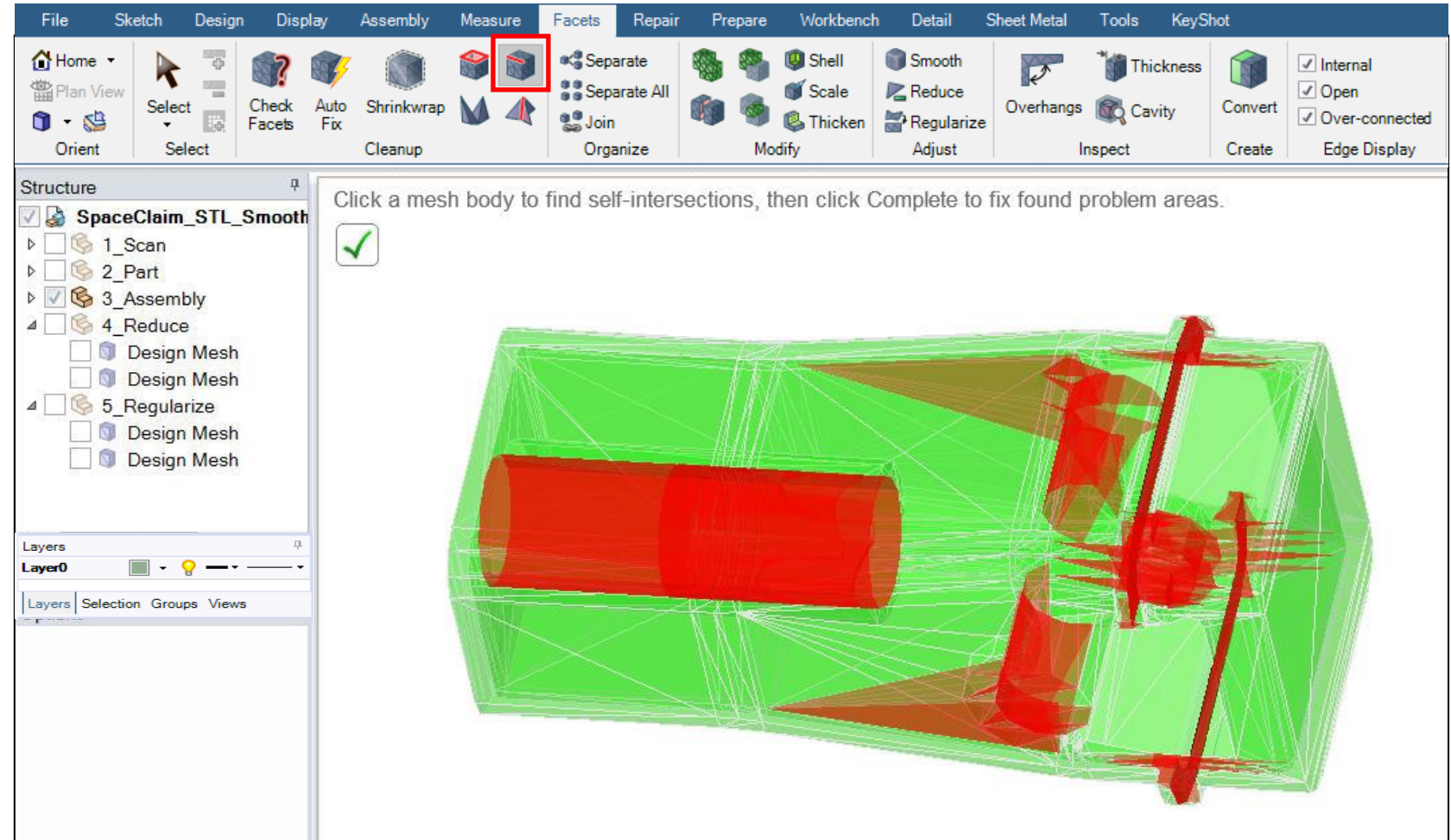


- Auto Fix uses straight faces to fill the holes irrespective of the shape of the geometry
- It is advised to double-check the problems fixed by Auto Fix

Handling STL Geometry (4)

- Intersections

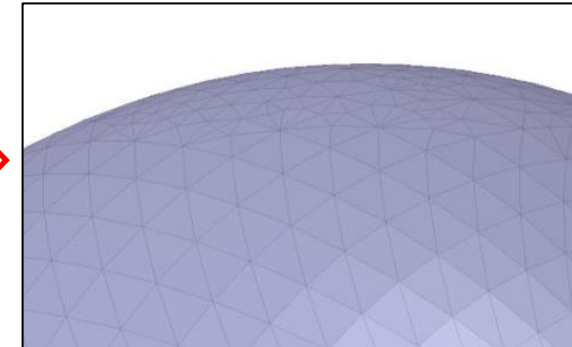
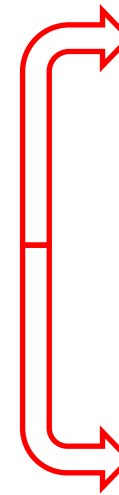
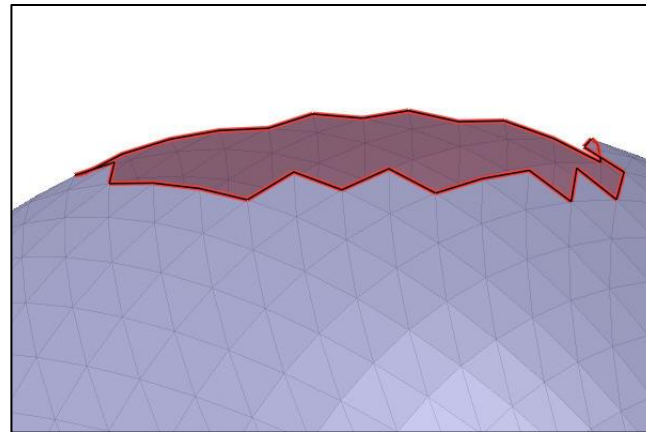
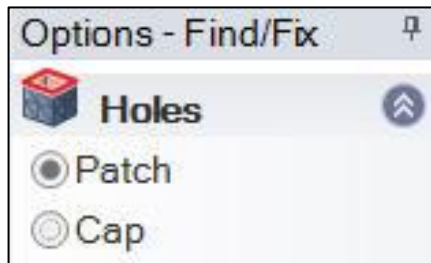
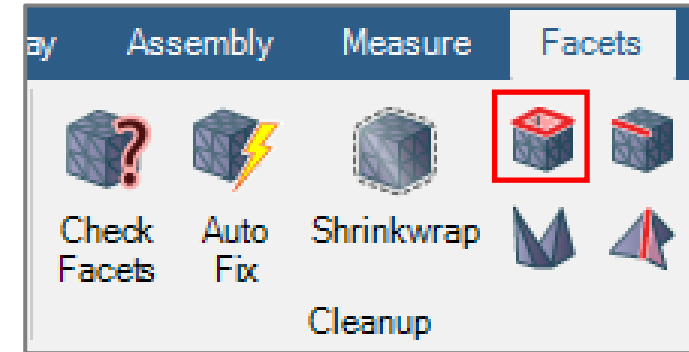
- Use the Intersections tool to highlight & fix the areas of self-intersection
- The model is also made transparent if any intersections are found



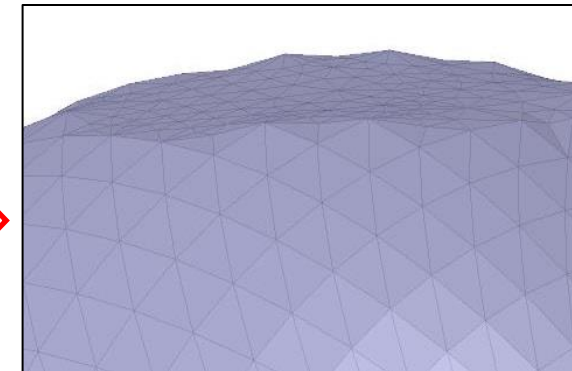
/ Handling STL Geometry (5)

- Holes
 - You can find and fix holes using the Holes tool
 - It works like other Find-and-Fix tools in the Repair tab
- Options- Find/Fix

Use “Faces” tool to remove remaining part of fillets



Patch



Cap

/ Handling STL Geometry (6)

- Fix Sharps

- Meshes may contain sharp spikes that need to be flattened

- Option- Find/Fix

- This applies to the angle between normals of facets that share an edge

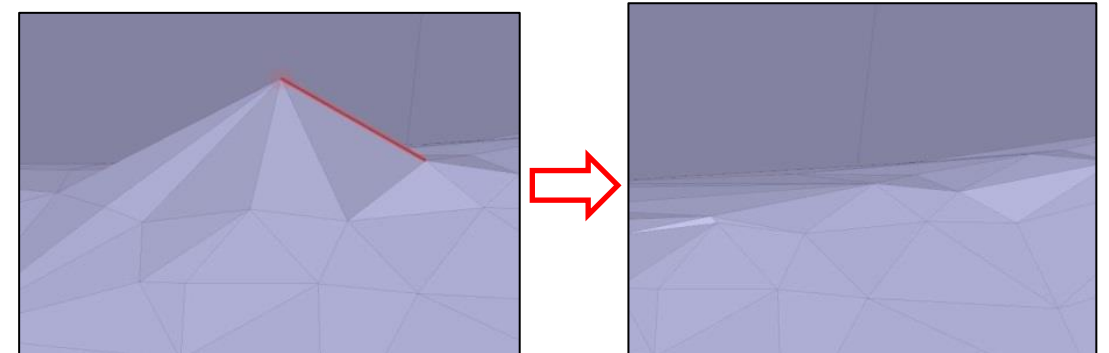
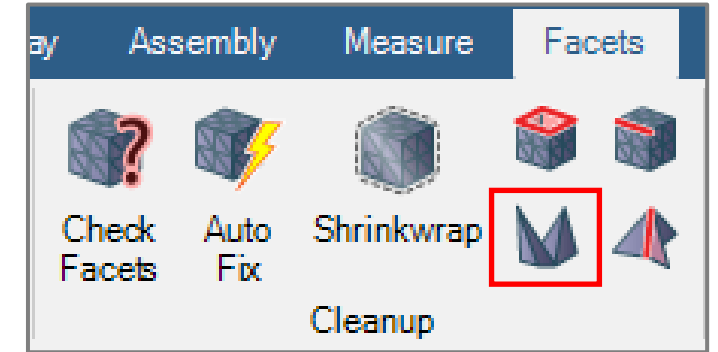
☒ Sharp edges

- This applies to the angle between normals of any two facets that share a vertex

☒ Sharp vertices


- Convexity options specify which type of sharps to look for

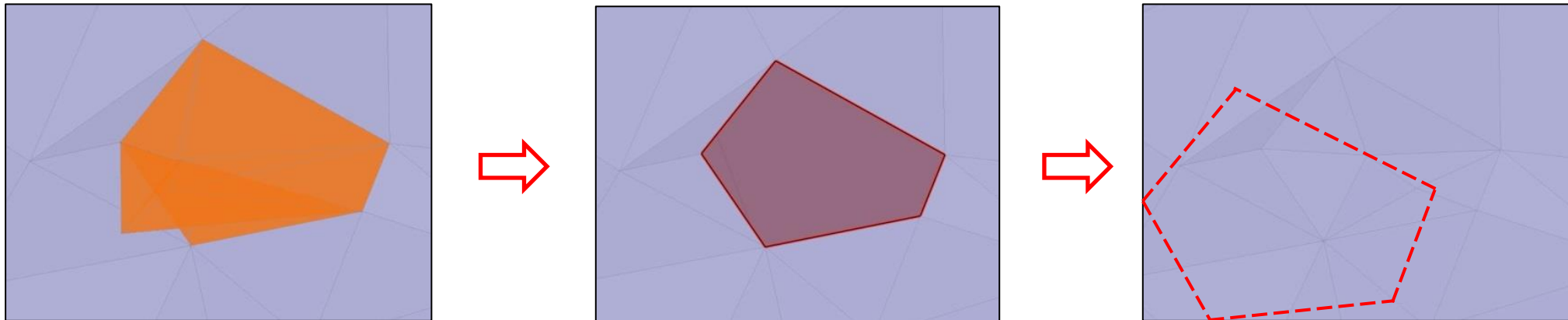
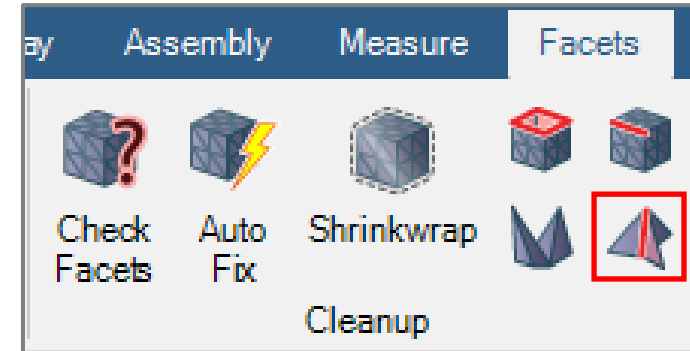
Convexity:
☒ Concave
☒ Convex



/ Handling STL Geometry (7)

- Over-connected/Non-manifold

- The multiple connected faces are highlighted
- If the problems are not fixed by the tools, then delete all the faces in local area and use fill holes  option to recreate the patch



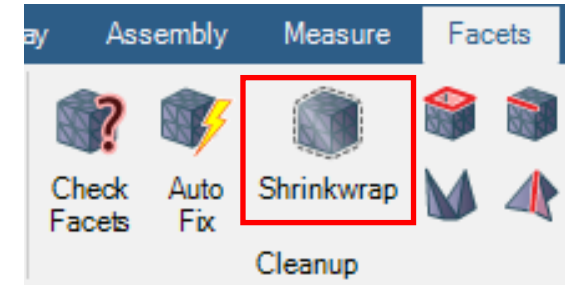
/ Handling STL Geometry (8)

- Shrinkwrap

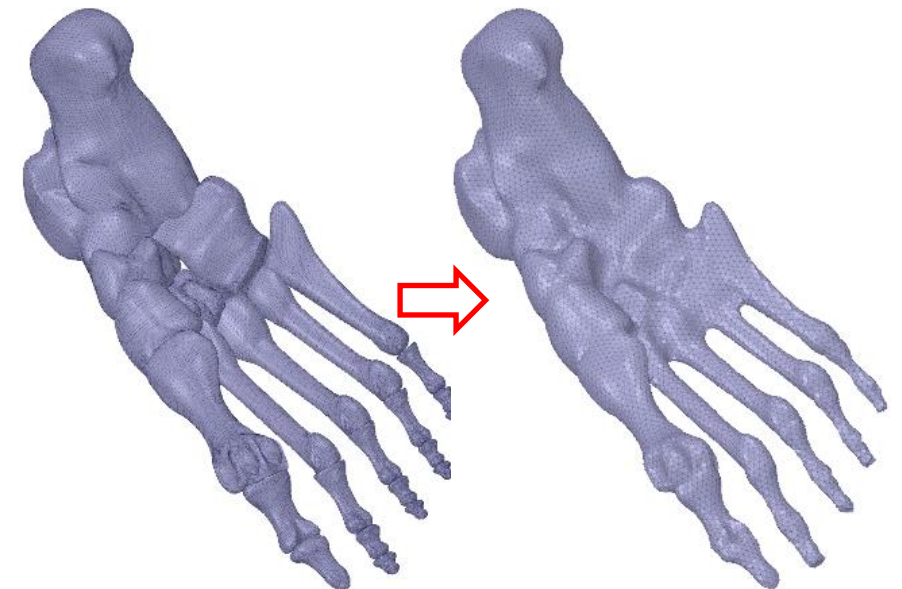
- Some faceted models contain gaps that will cause problems for 3D Printing. For example, assemblies have multiple parts fastened together but there are small gaps between the component parts. The Shrinkwrap tool allows you to create a single, faceted model by closing gaps that can be sent to a 3D printer

- Options Panel

- Gap size: Maximum distance between meshes that will be considered a gap
- Keep original bodies: When checked ON, the original faceted bodies will NOT be removed after the Shrinkwrap is created
- Preserve features: When checked ON, you need to set an Angle threshold
- Angle Threshold: Angle, in degrees, that determines the smoothness of the shrinkwrap



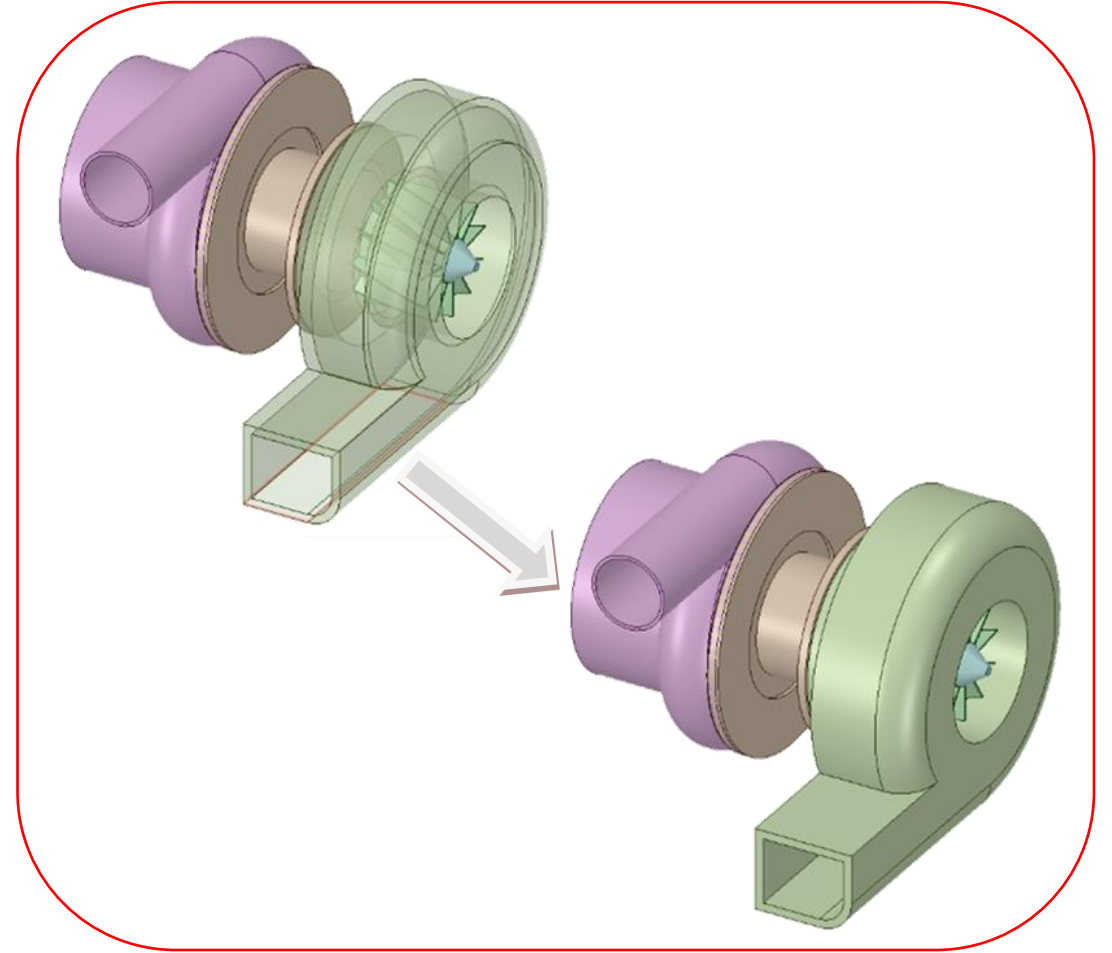
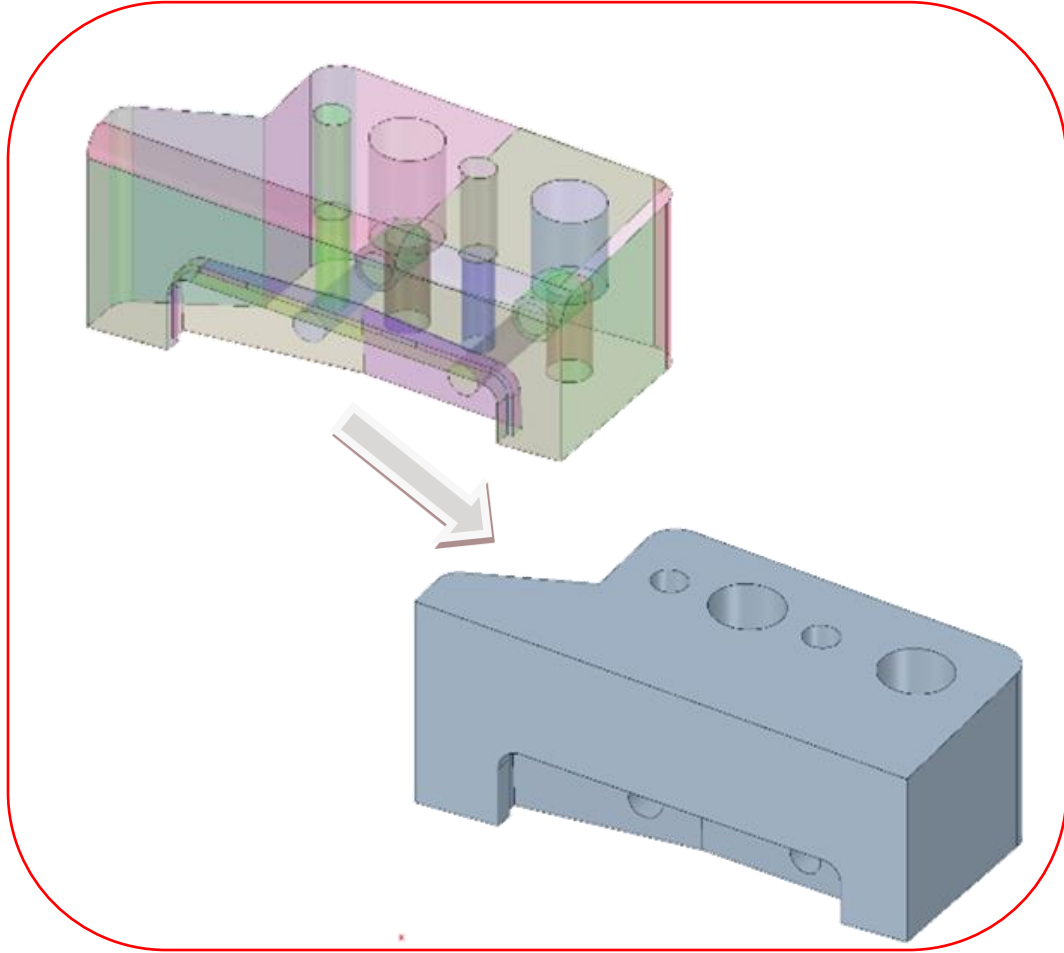
The foot model used in this example has small gaps between the bones of the foot



/ Summary

- SpaceClaim has a large number of repair tools to edit and repair imported CAD
- In the repair tab, there are tools to solidify non watertight geometries in addition to fixing edges and faces
- Other tools are available in the Prepare tab to detect and repair the interference, to remove the rounds and faces and to detect bad faces and edges
- In the Tools tab, the 'Reverse Engineering' group allows you to import an STL file and repair it in various ways
- STL mesh can be cleaned up using the Cleanup group in the Facets tab

Workshop 3.1 Repairing Geometry





End of presentation