Introduction to Ansys SpaceClaim

Workshop 3.1: 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.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

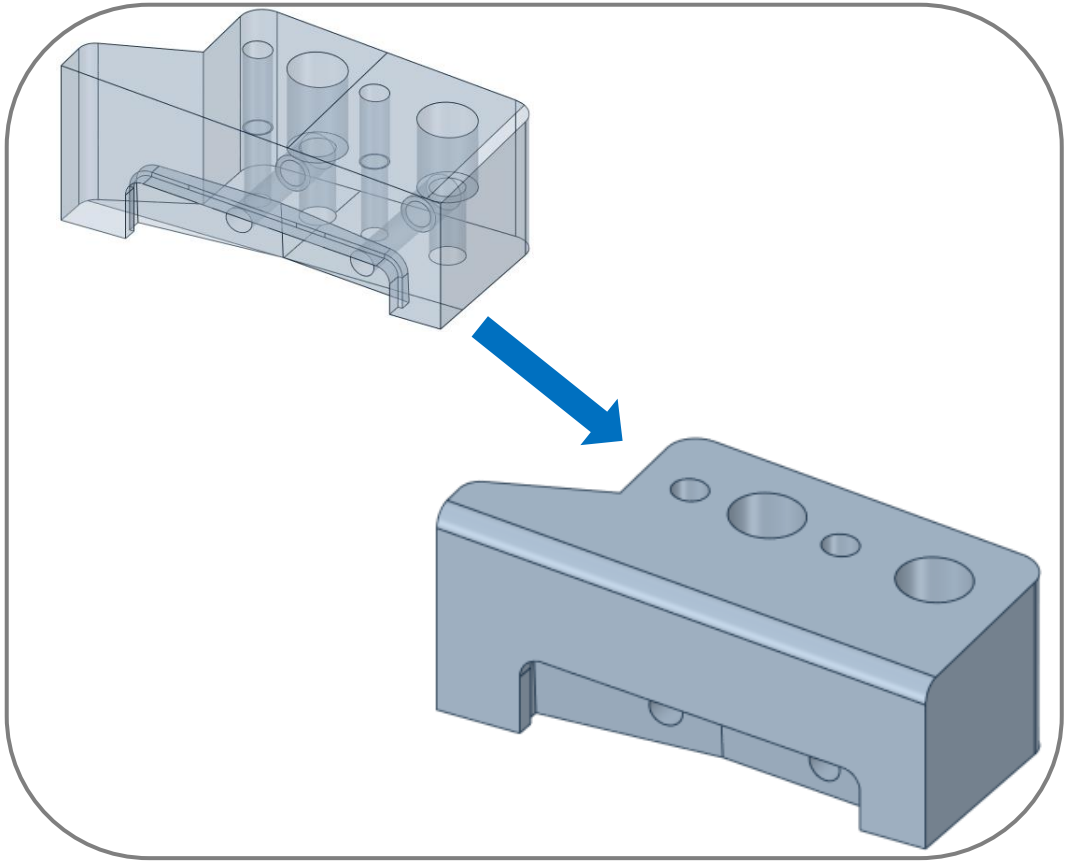
Key Learning Points

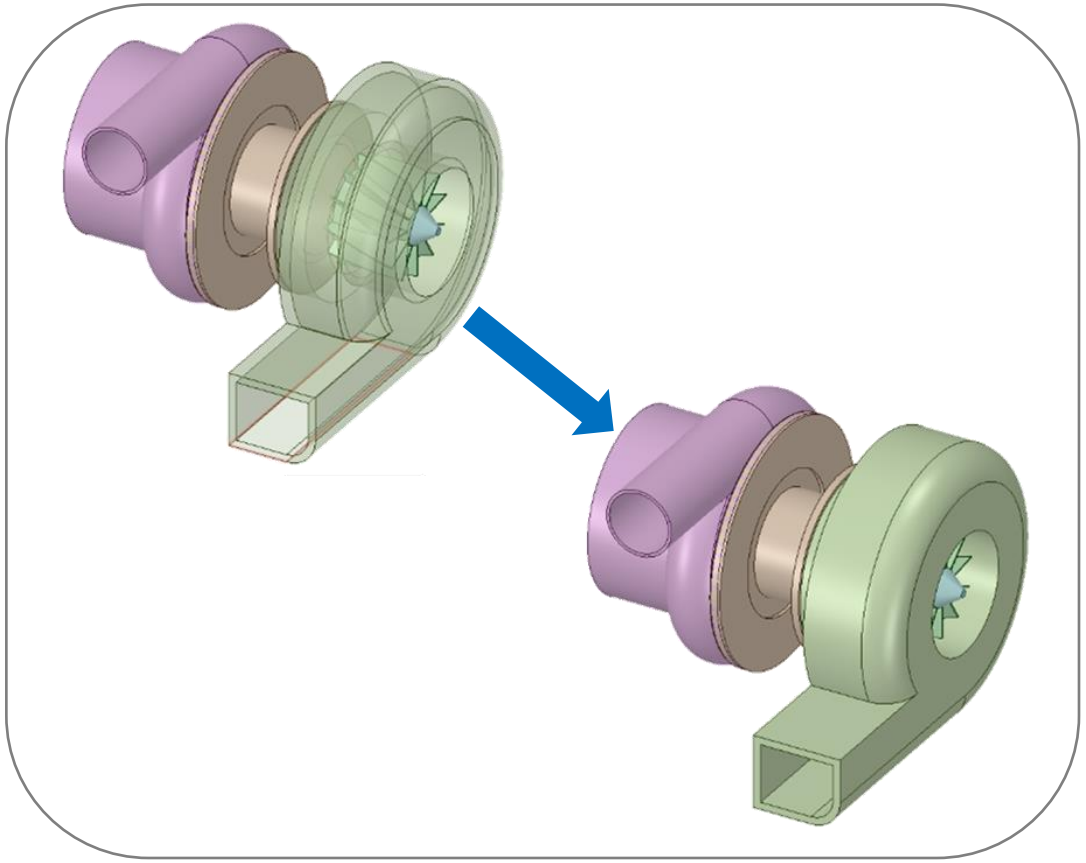
Completion of this workshop will help to gain an understanding of:

- How to repair poor geometry
- How to implement best practices when working in SpaceClaim
- Importing CAD geometry
- Specialized tools available in SpaceClaim

Objectives

- Geometry repairs that lead to solidification of a problematic geometry
- Application of saved “groups”
- Interference detection and removal between parts

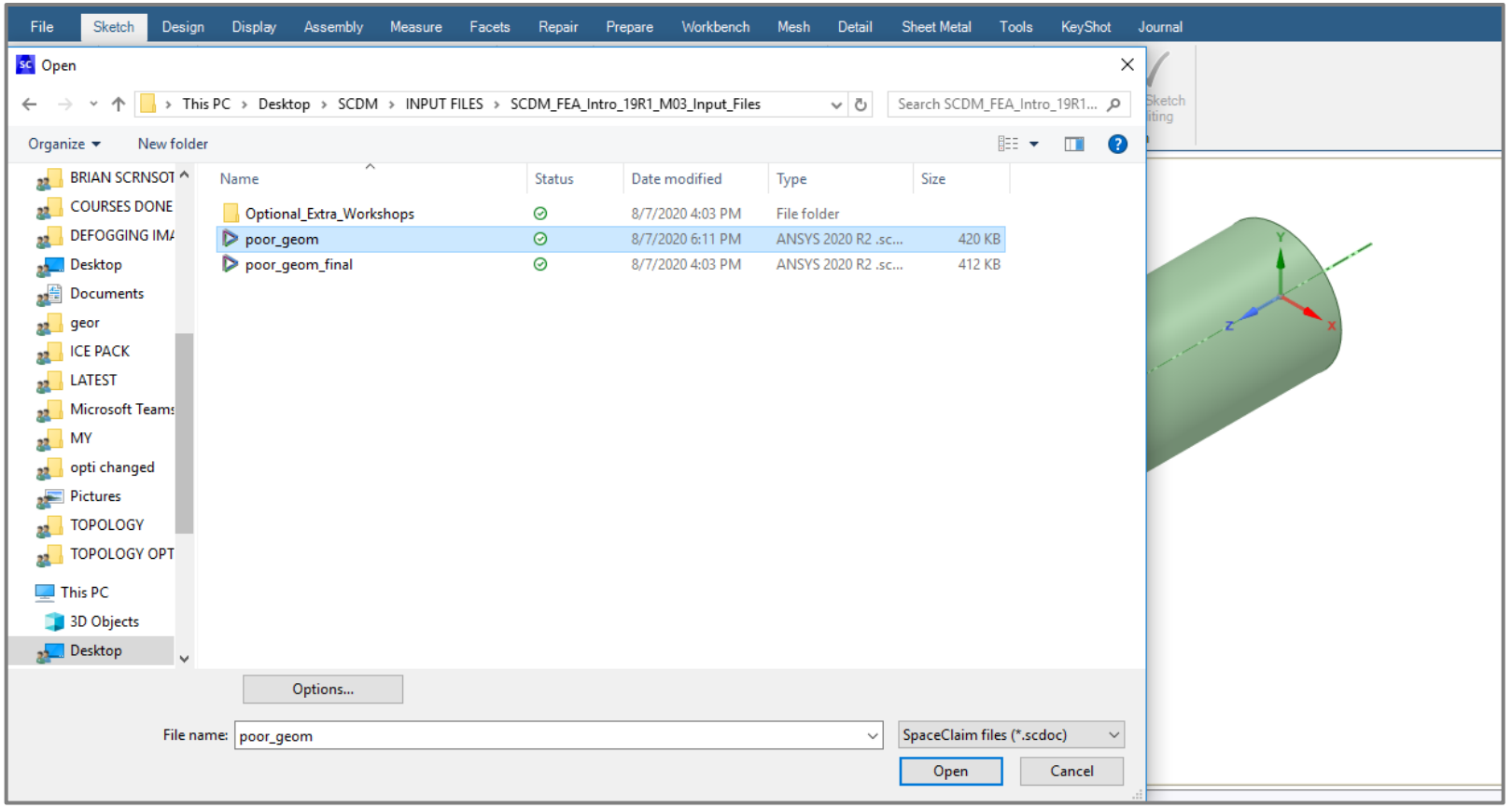




Repairing poor geometry

Launch SpaceClaim and open the file “poor_geom.scdoc”

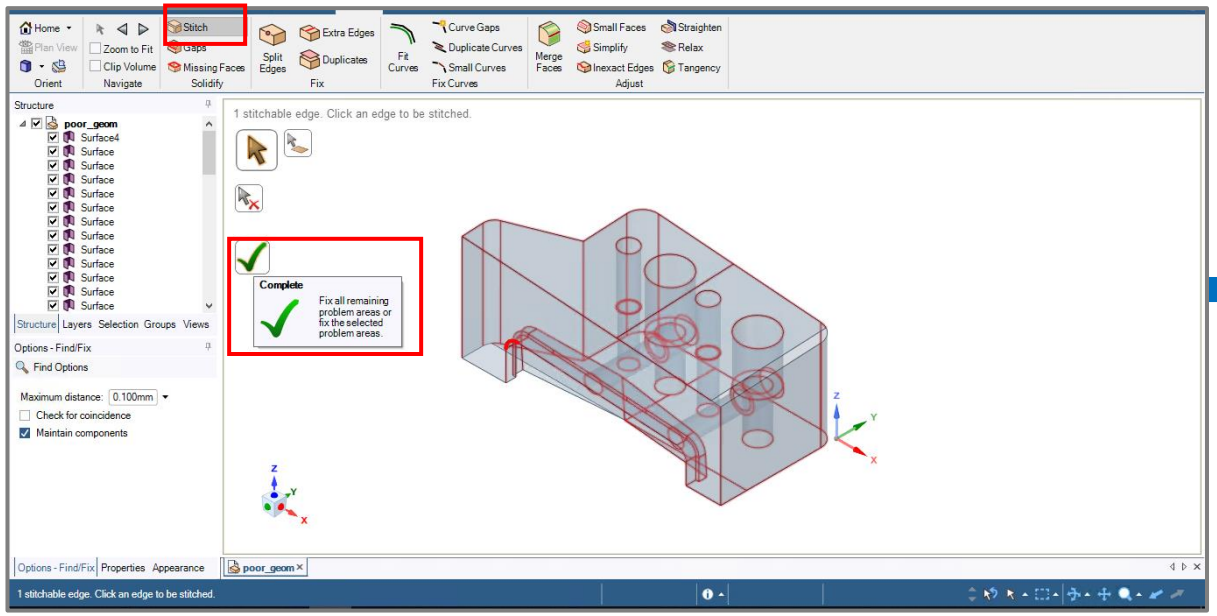
- Either “SpaceClaim Files” or “All Files” must be selected for the file to be visible
- Note that the geometry is composed of many different surface objects

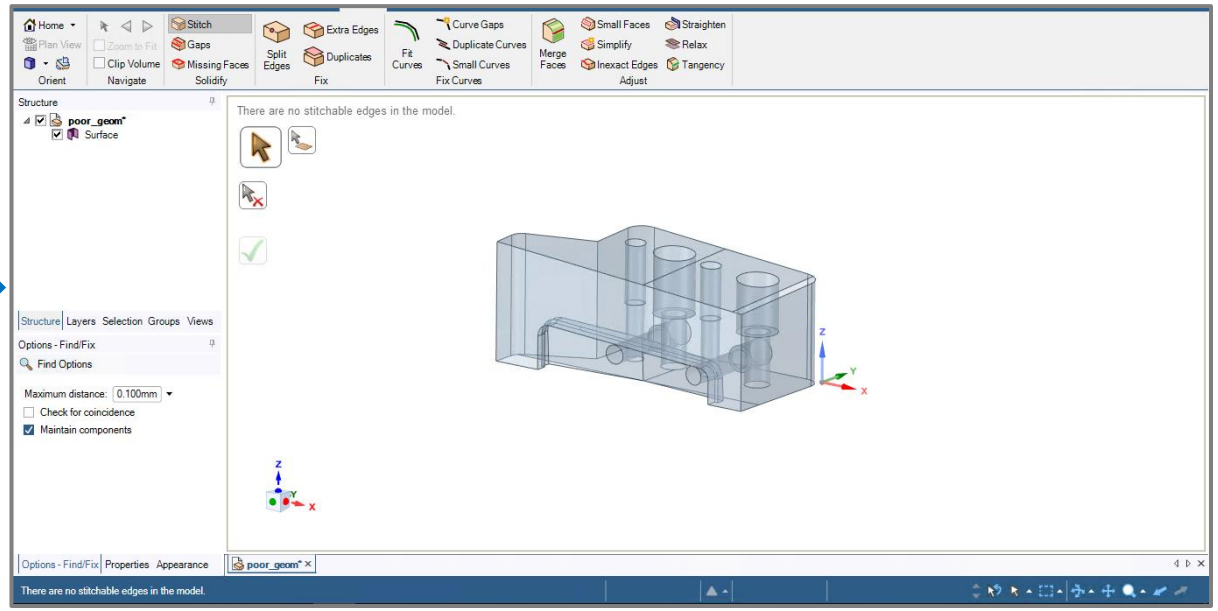


Common issue: stitching together surfaces

Selecting “**Repair->Stitch**” highlights edges of adjoining surfaces

- Click the green “check” to complete the stitch
- Note that the resulting geometry is a single surface object

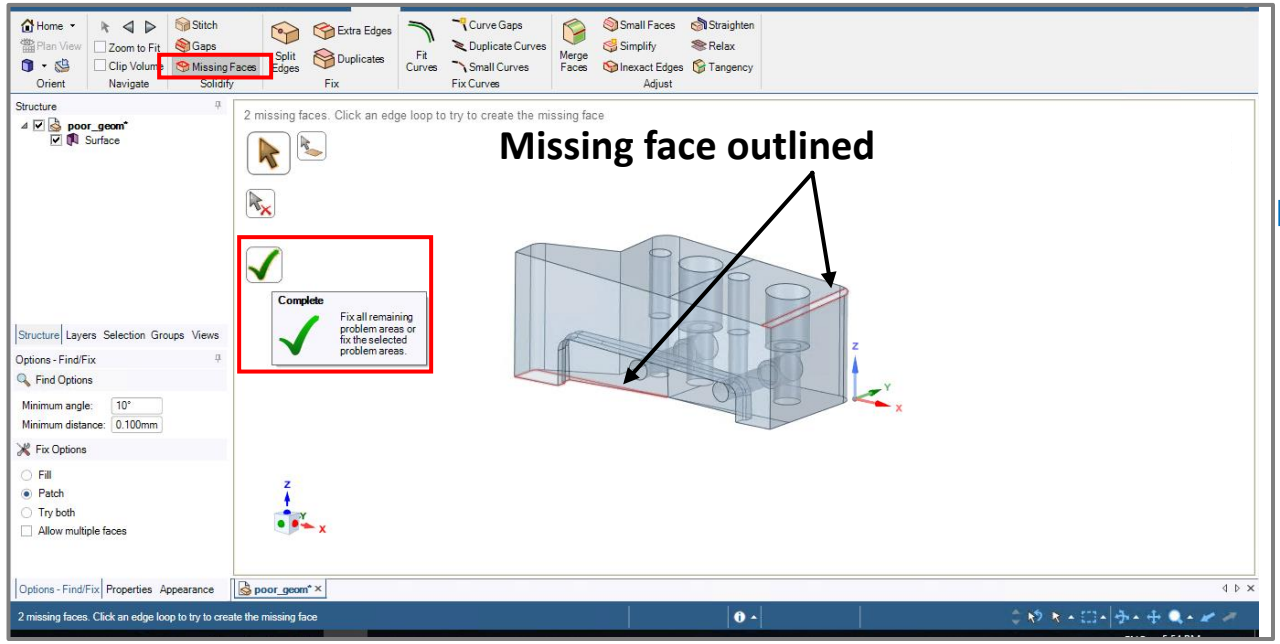


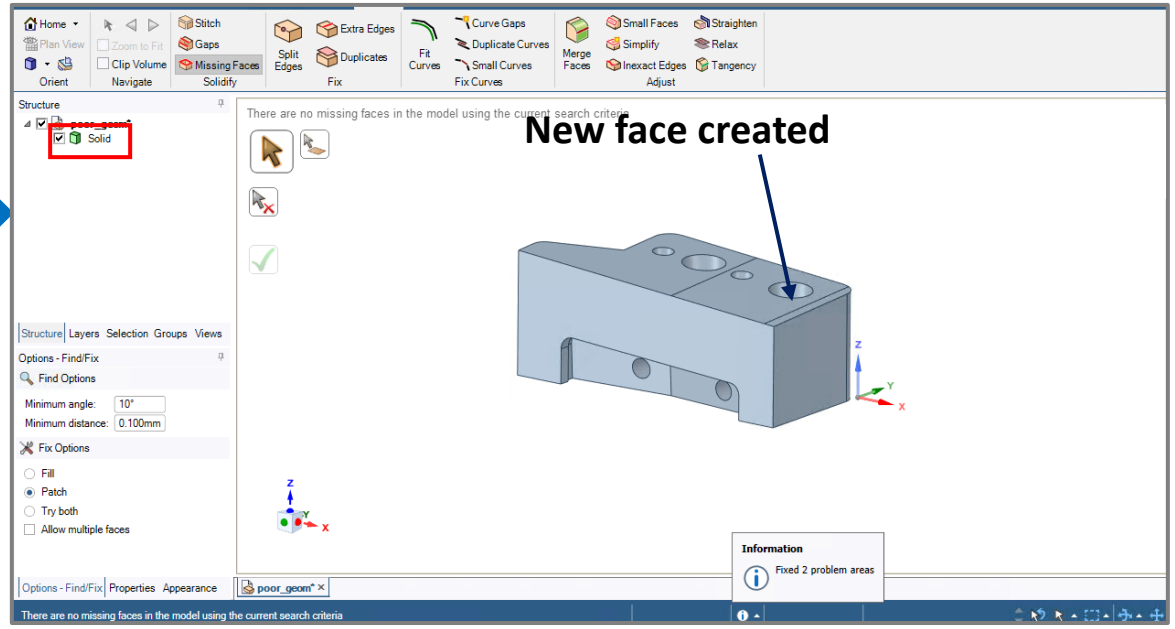


Common issue: repairing missing faces

Selecting “**Repair->Missing Faces**” highlights missing faces where edges are not joined

- Click the green “check” to complete the repair
- Note that a new solid has been created

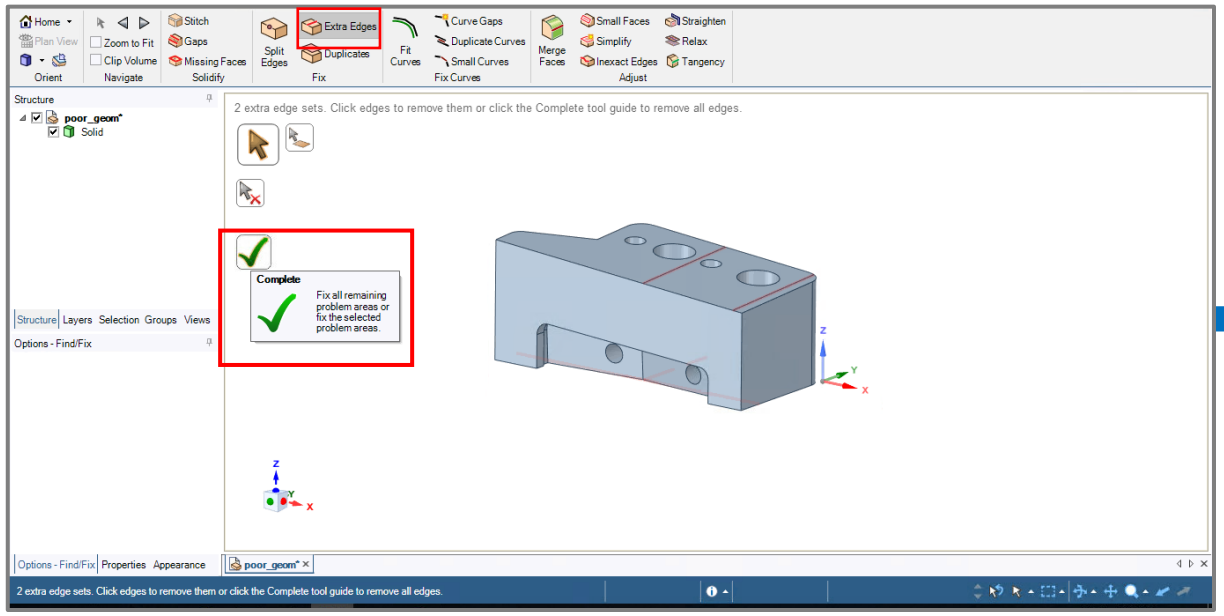


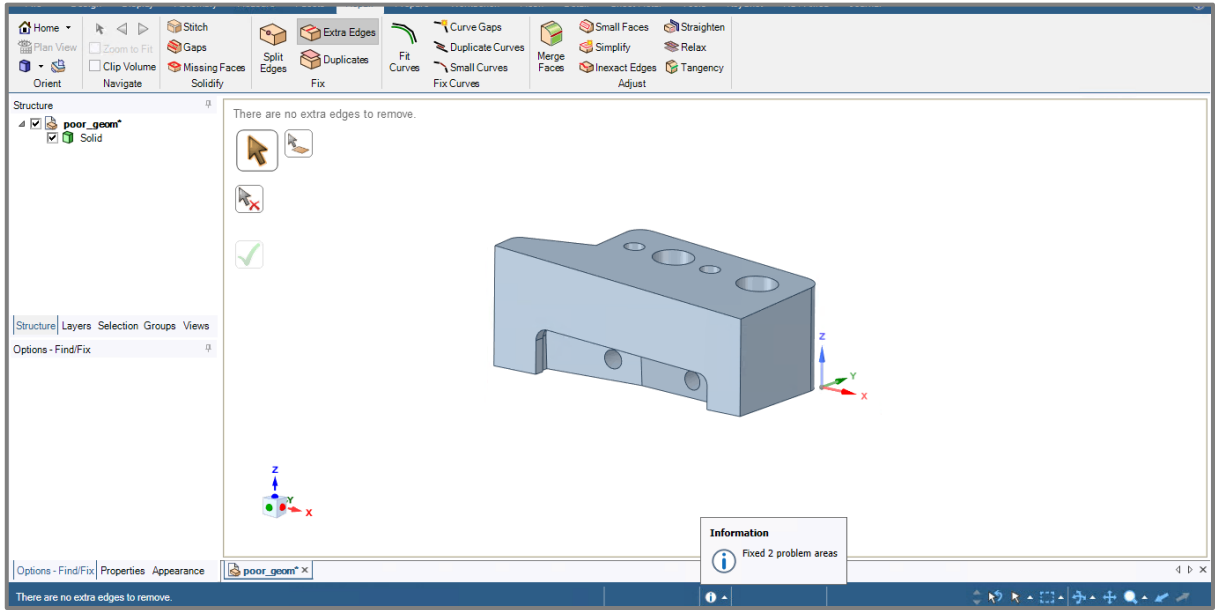


Common issue: merging faces

Select “**Repair->Extra Edges**” to try to delete extra edges

- The thin face created by the repair is unnecessary: merge it with the larger neighbour face
- The new face covers the top opening

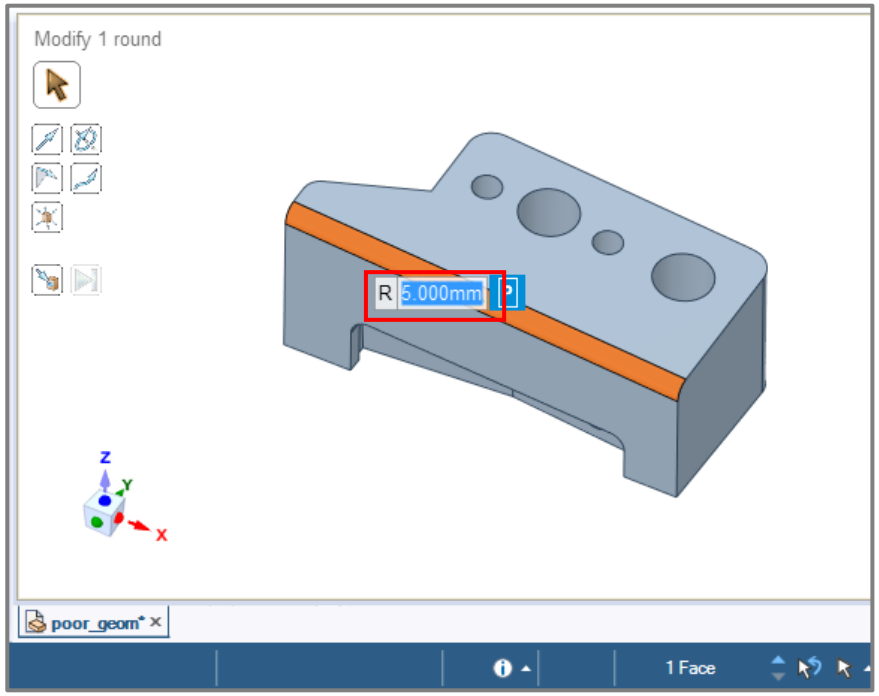


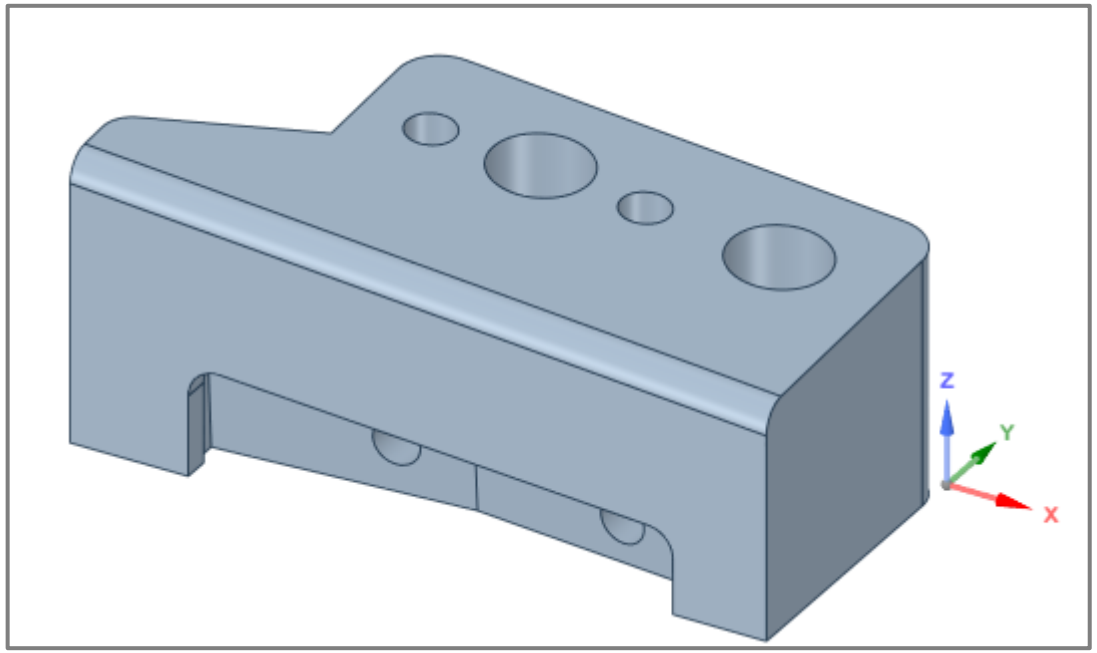


Creating a Round

Select “**Design->Pull**” and select the edge shown here to create a round

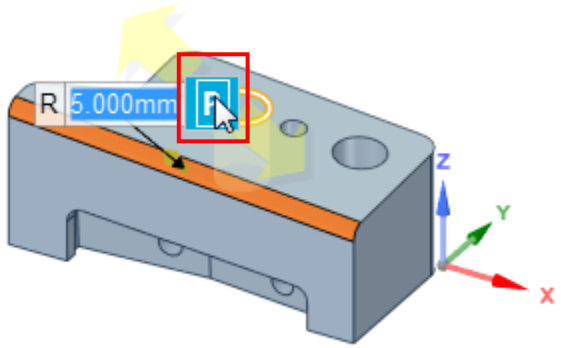
- Type in the radius 5 mm

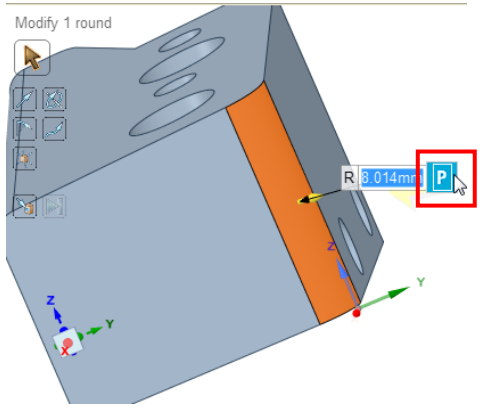


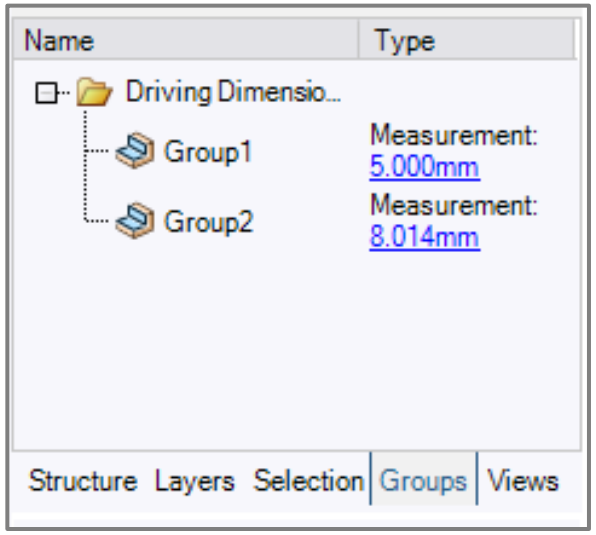


Best practice: Creating groups

- Select the round and click ‘P’ key to create a group with the round radius
- Repeat the same operation at the other-side round as shown below



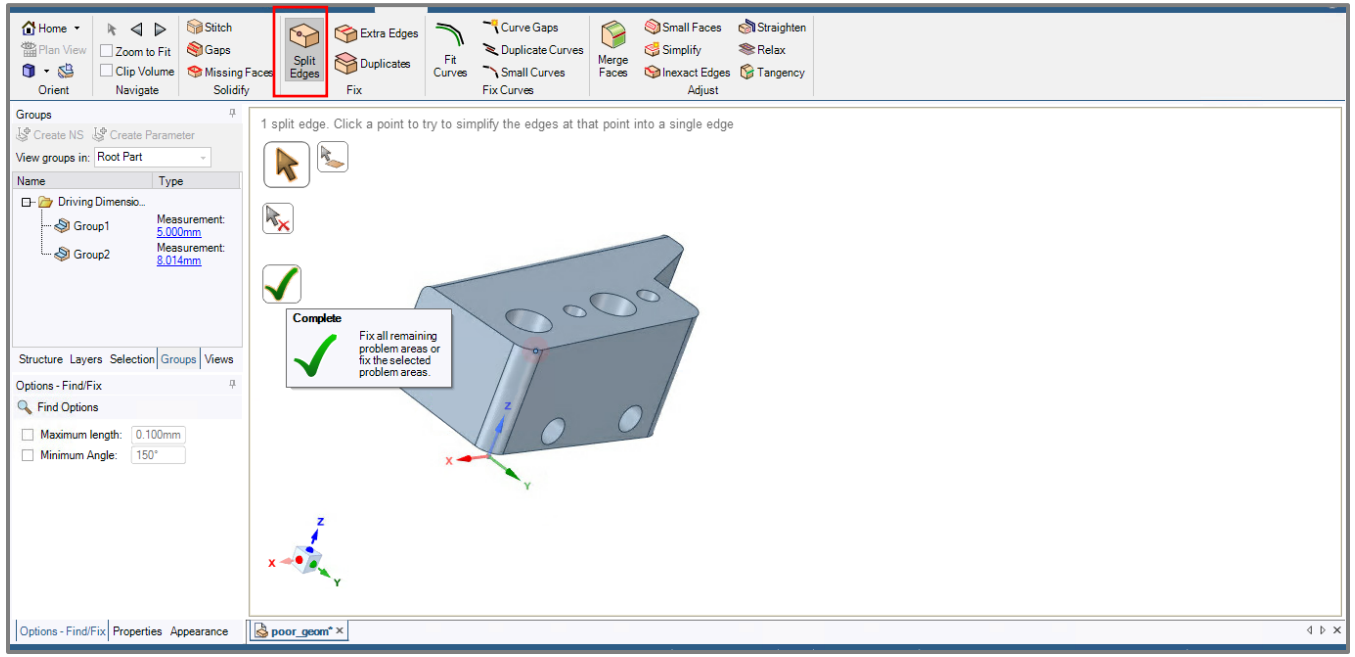


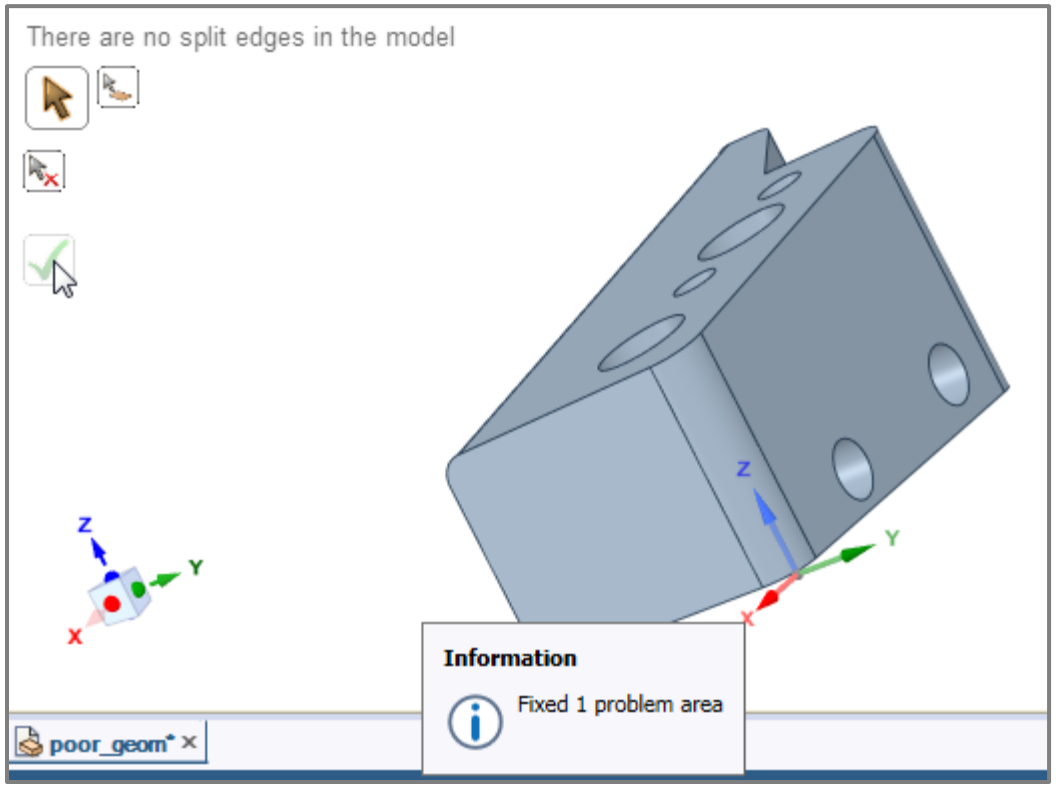


Best practice: repairing split and inexact edges

Select “**Repair->Split Edges**”… SCDM will find 1 area to fix

- Sometimes, try “**Repair->Inexact Edges**” before Split Edges if SCDM fails to fix them
- Clicking many times on the “Complete” can help fixing more problems

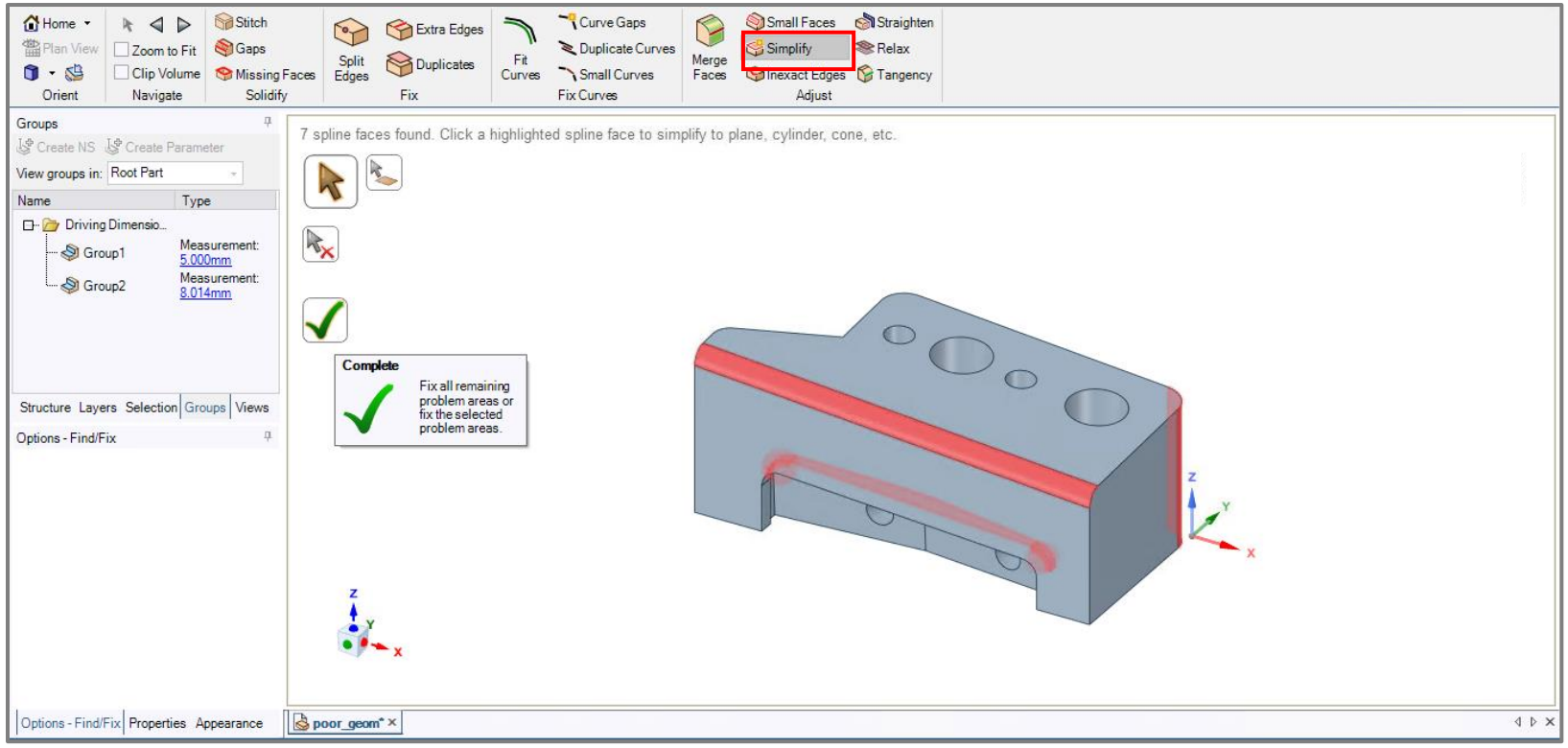




Simplifying geometry

Selecting "**Repair->Simplify**" reconstructs geometry with simpler topology

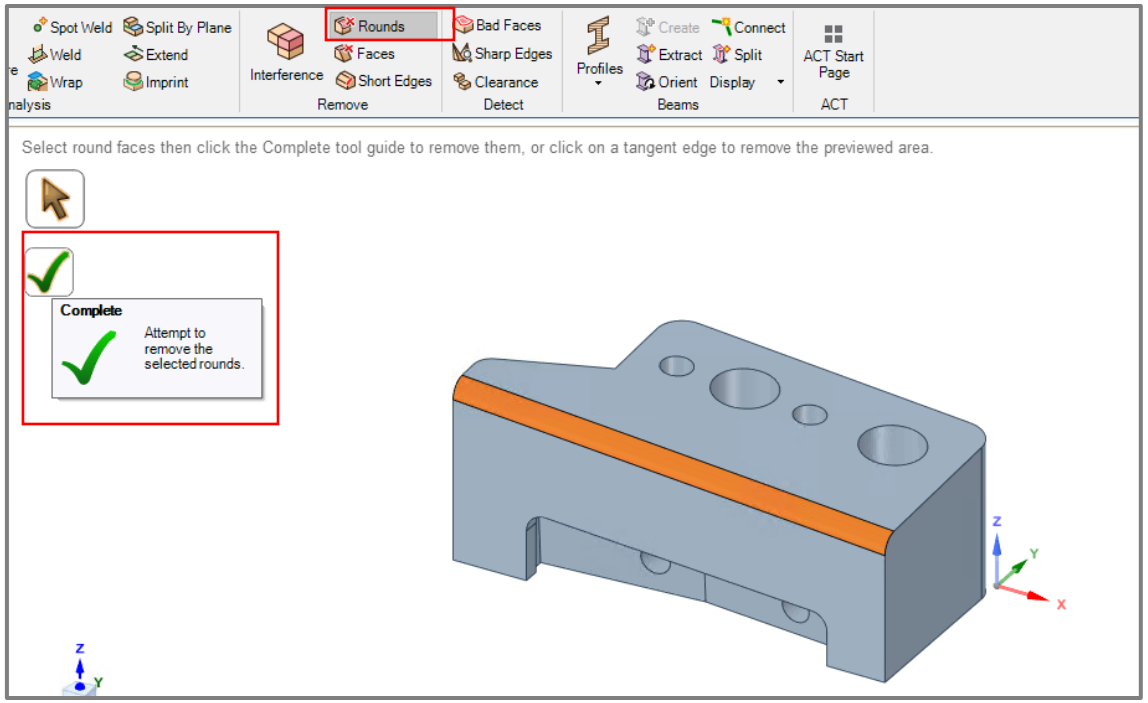
- Select "**Repair->Simplify**" on the rounded edge to simplify its shape
- NOTE: "**Repair->Simplify**" can be performed on the whole model at once if desired

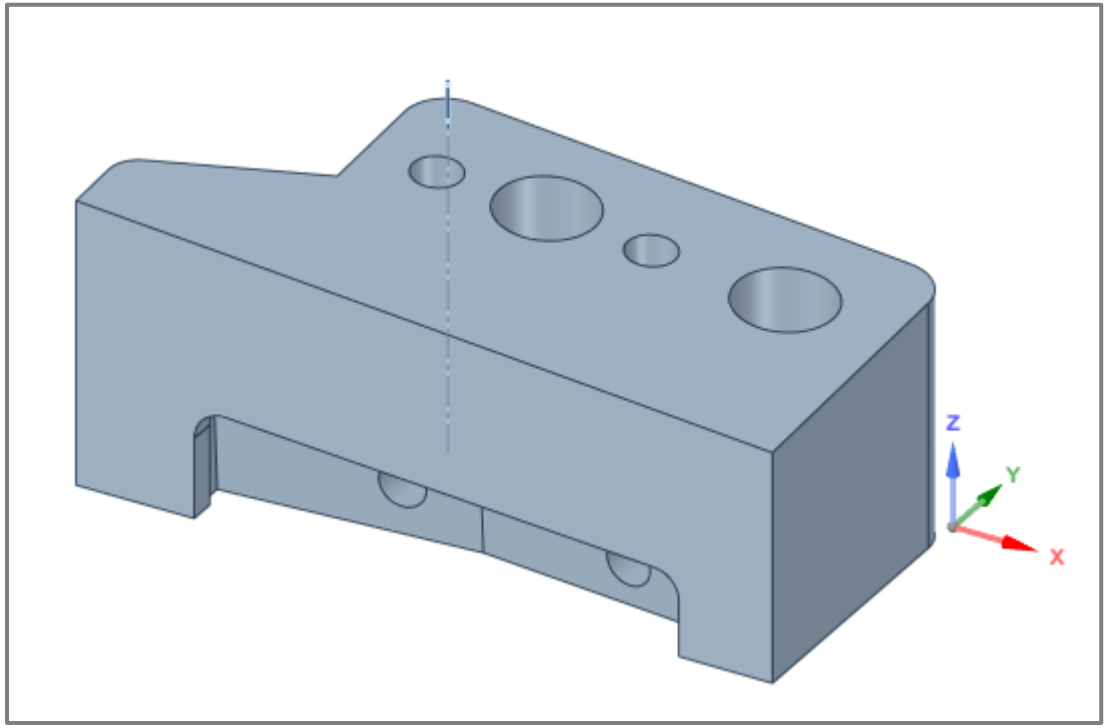


Specialized tool: removing rounds

Selecting “**Prepare->Rounds**” will remove rounds from edges

- Only object recognized as rounds are selectable
- The removed rounds are stored under the “Groups” tab, near the structure tree

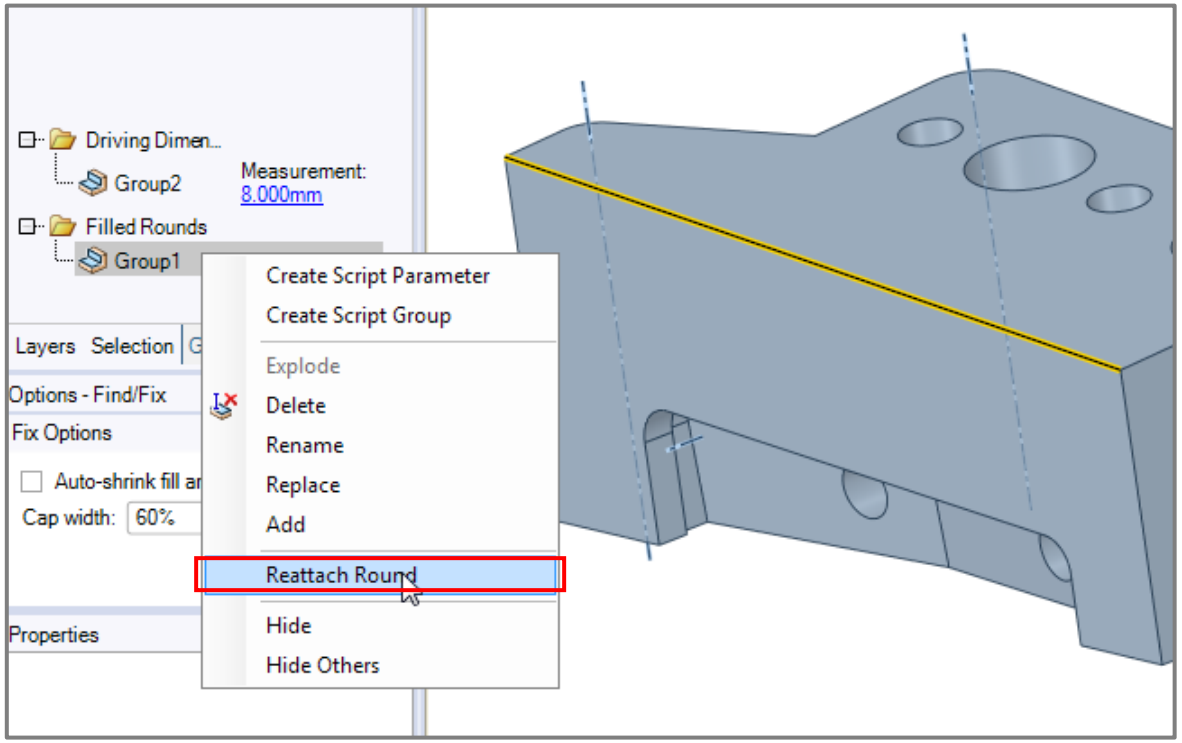


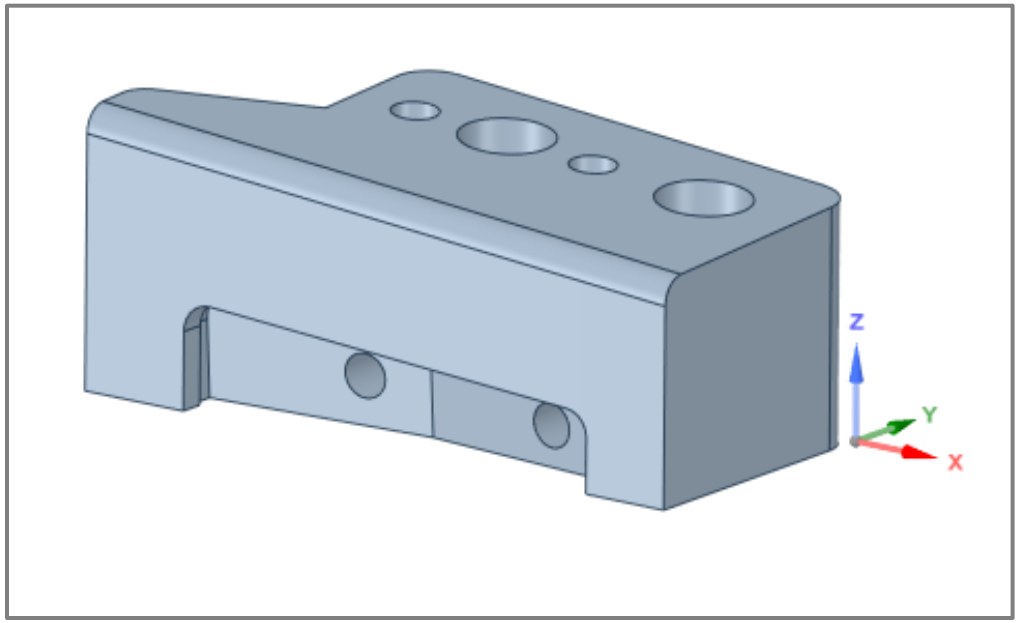


Specialized tool: reapplying rounds from groups

Rounds can be reapplied from the “Groups” tab where previously stored

- Right-click “Round Group1” and “Reattach Round”
  - The “Groups” tab is located near the structure tree

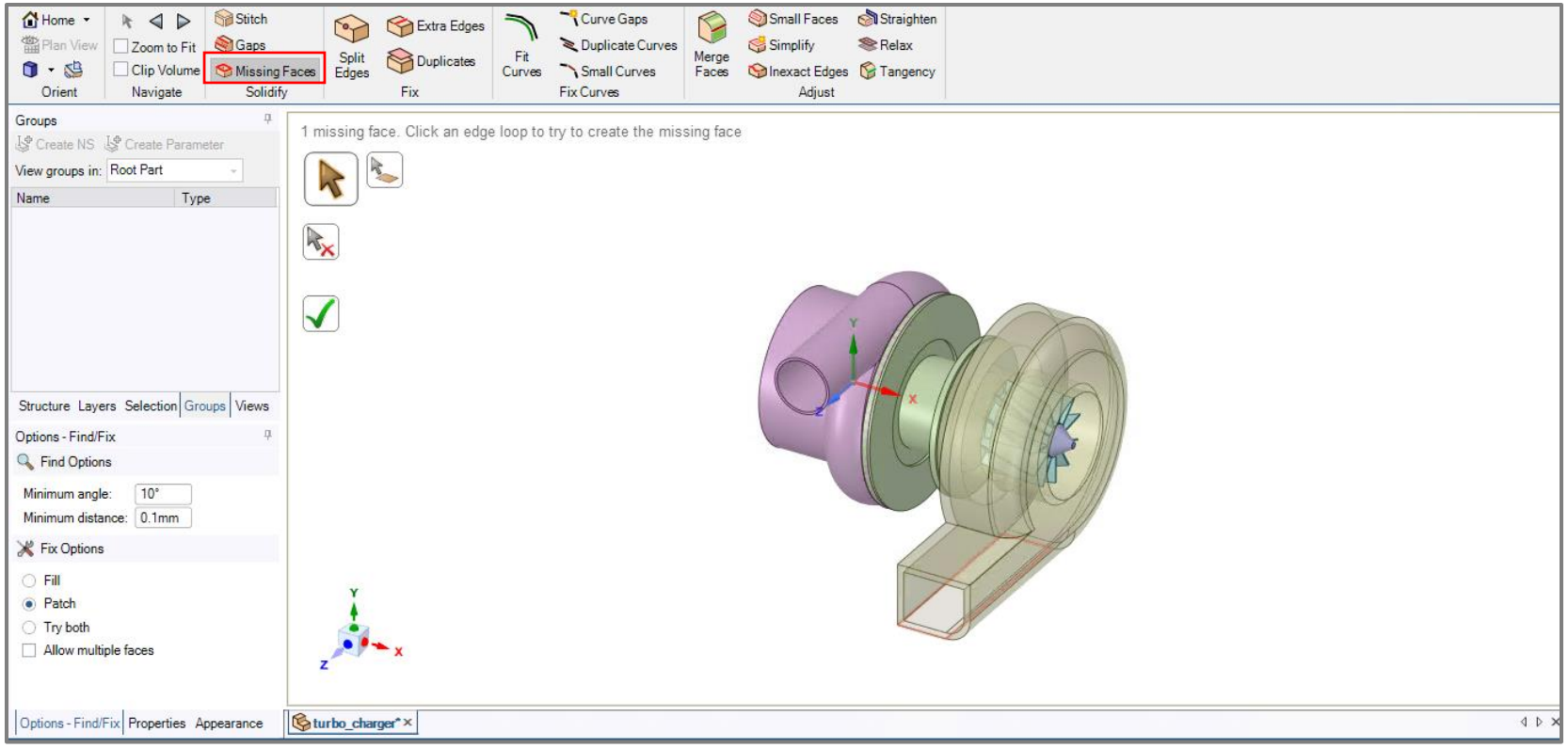




CAD Import and Clean-up

Import the Parasolid file “turbo_charger.x_t” (only visible if “Parasolid Files” or “All Files” is selected from the file type list)

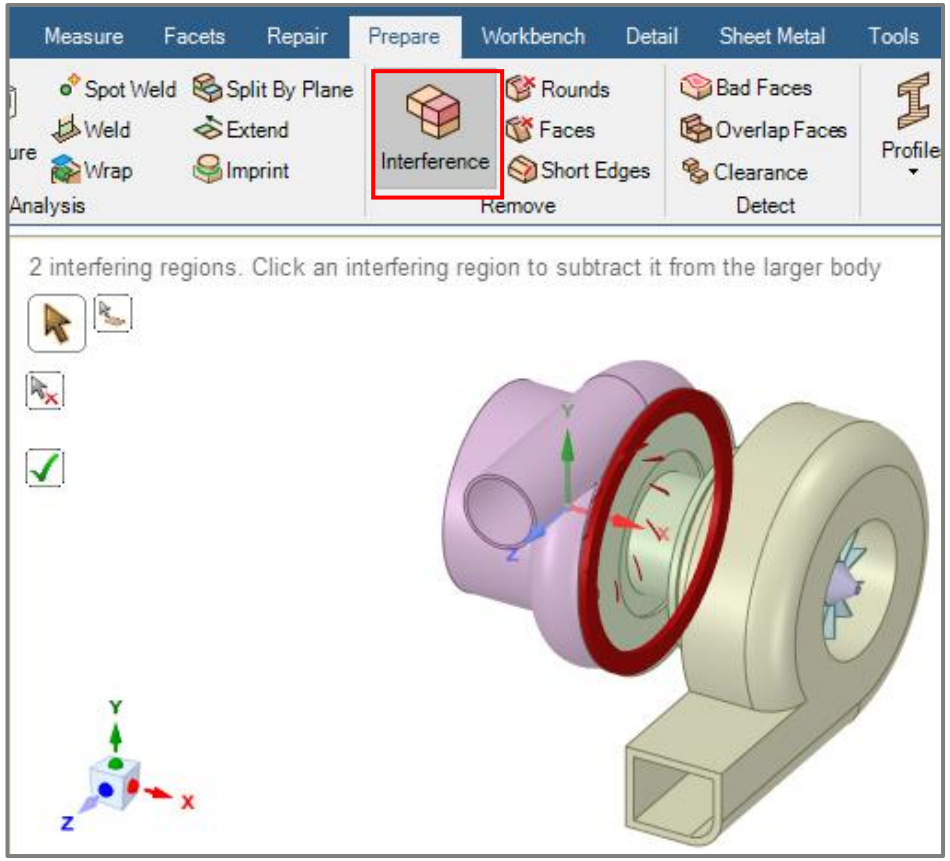
- Solidify the geometry with “**Repair->Stitch**” and “**Repair->Missing Faces**”



Specialized tool: interference repair

"**Prepare->Interference**" will detect and remove interference caused by different parts of the geometry overlapping with one another

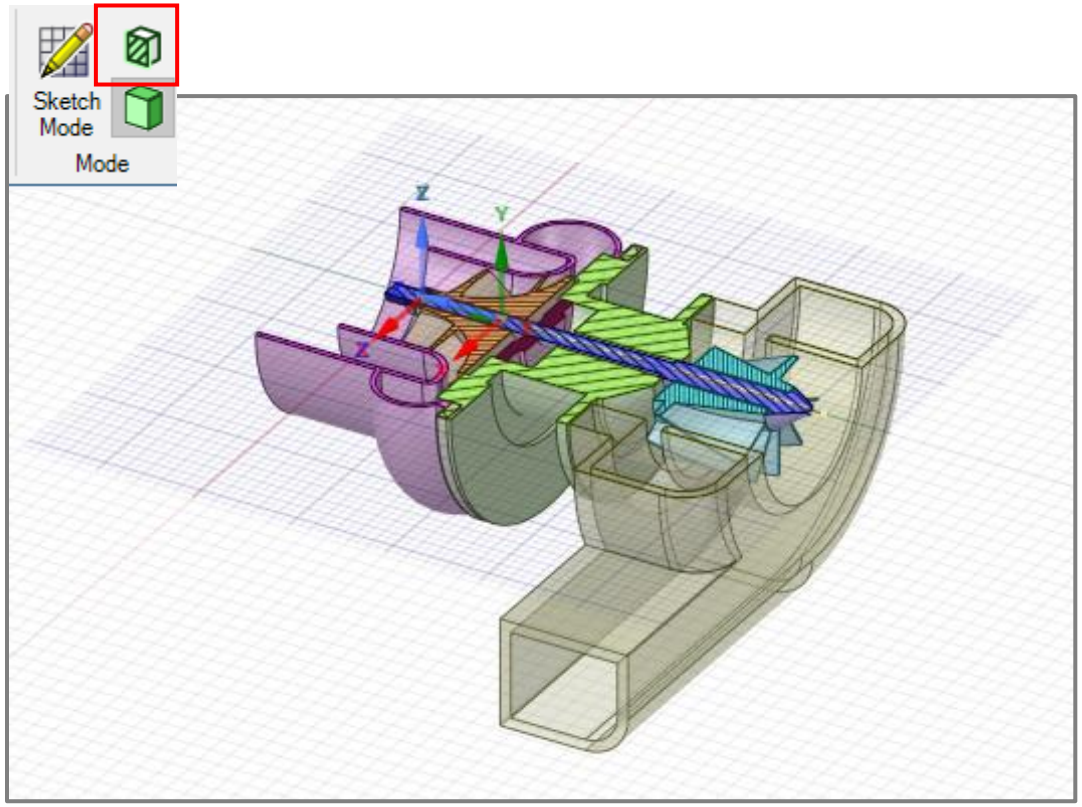
- Select "**Prepare->Interference**" to view two overlapping area

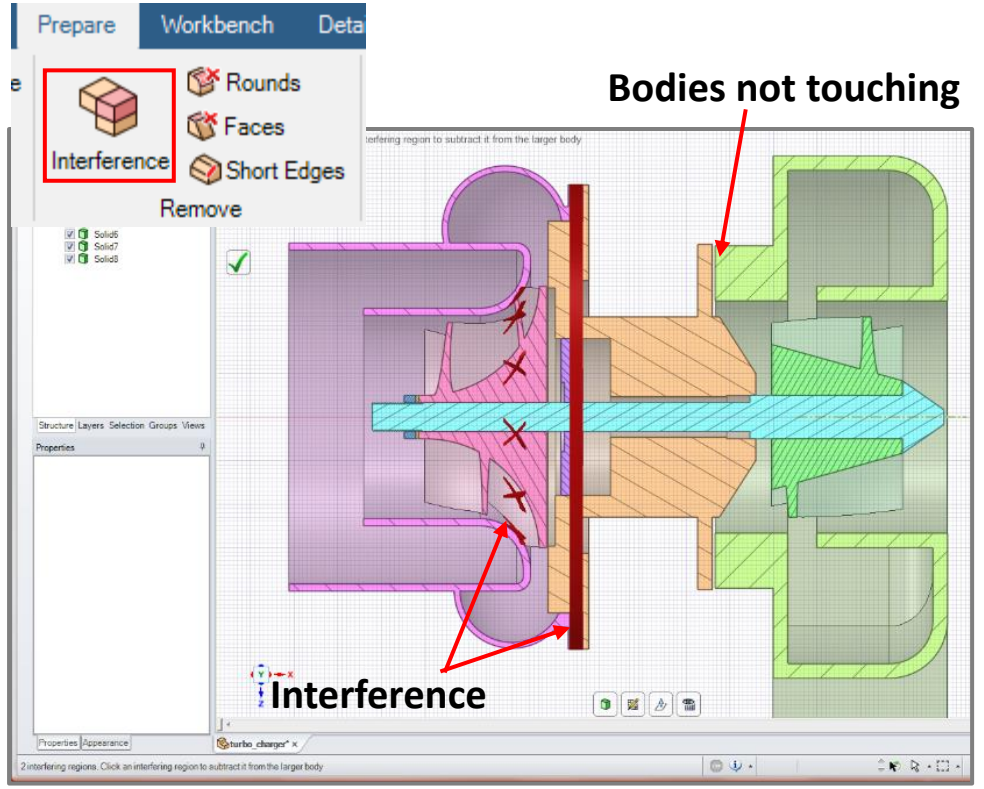


Section mode

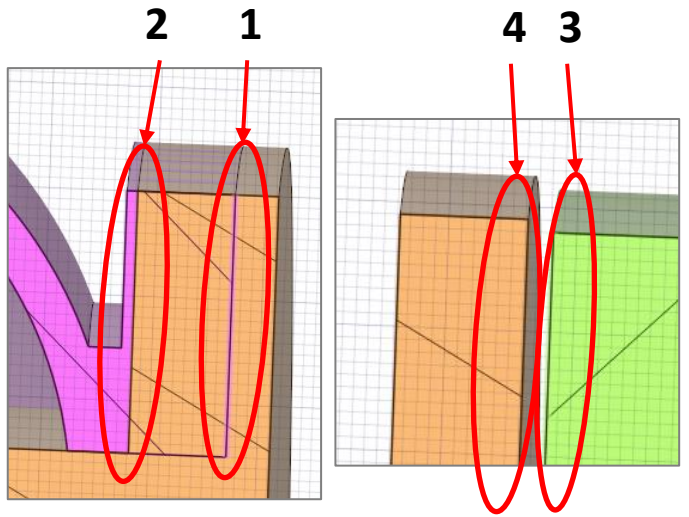
Examine and repair the assembly in section mode

- Select the central axis, enter section mode, and select “Plan View”
- Detect interference with “**Prepare->Interference**” (but do not repair – we will do this manually)
- Note the presence of the small gap between two of the components



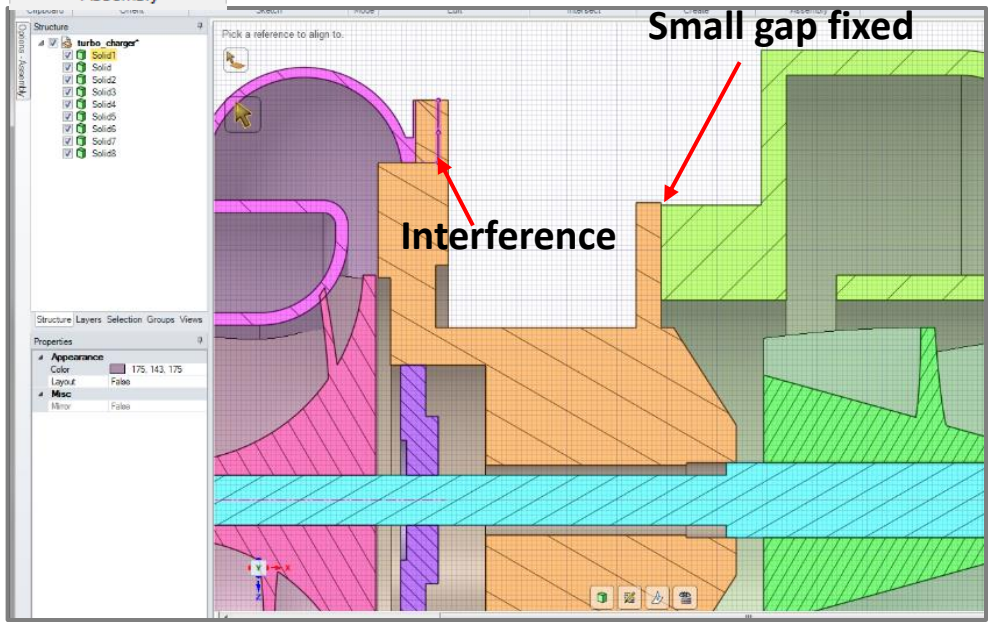


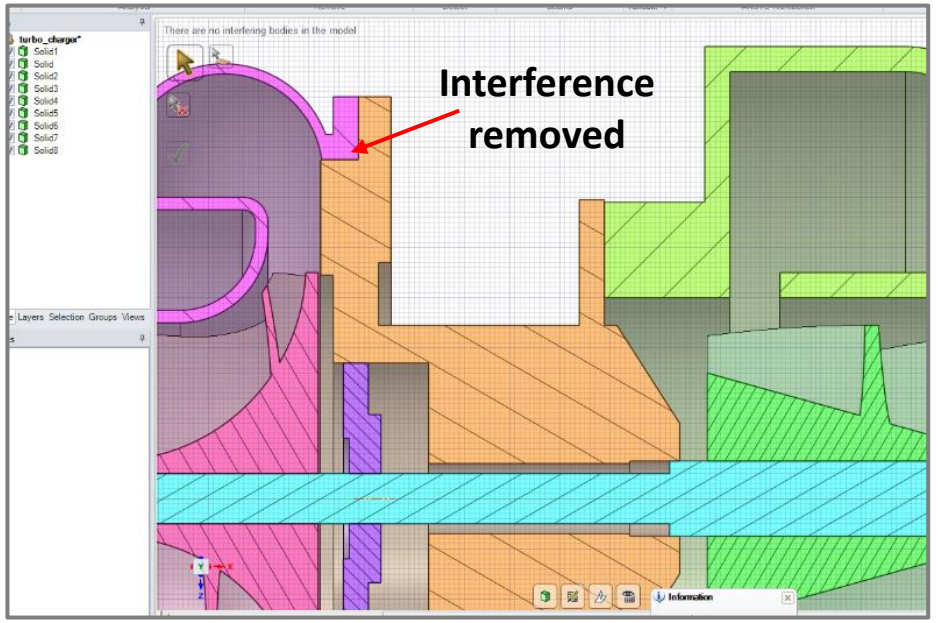
Removing the interference (1)



Align the objects to remove the interference and the small gap

- Select “Assembly->Align” click in order on edges: 1, 2, 3, 4
  - Remember that edges in section mode are actually faces in 3D
- Once complete, note that “Prepare->Interference” finds no problem areas





End of presentation