

## Module 04: FEA Modeling

Release 2021 R1

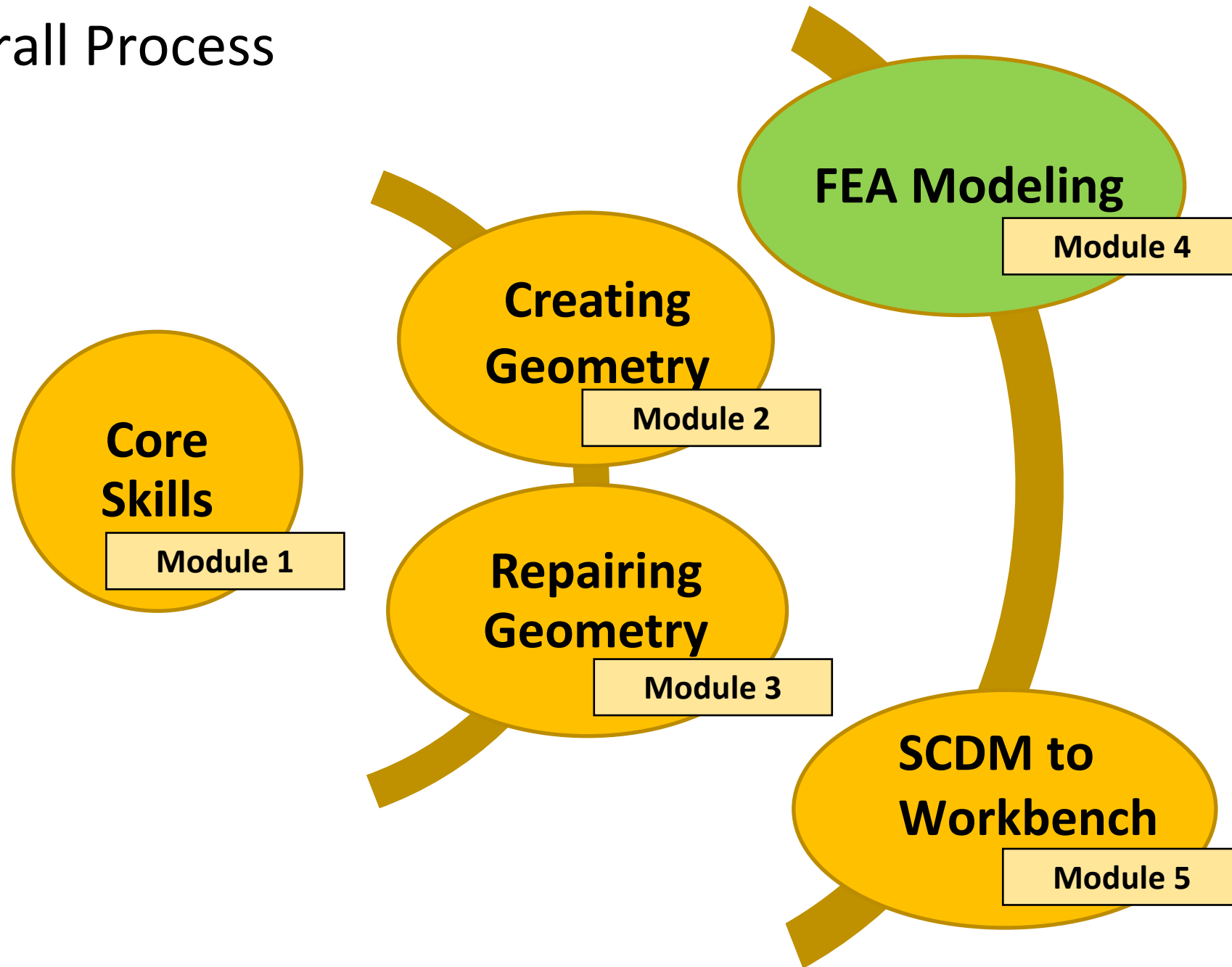
Please note:

- 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

- How can we create spot welds in SpaceClaim
- Discovering the weld feature
- How to create midsurfaces in SpaceClaim
- Discovering the Beam Extraction in SpaceClaim
- Discovering the SpaceClaim toolbox in the App Store

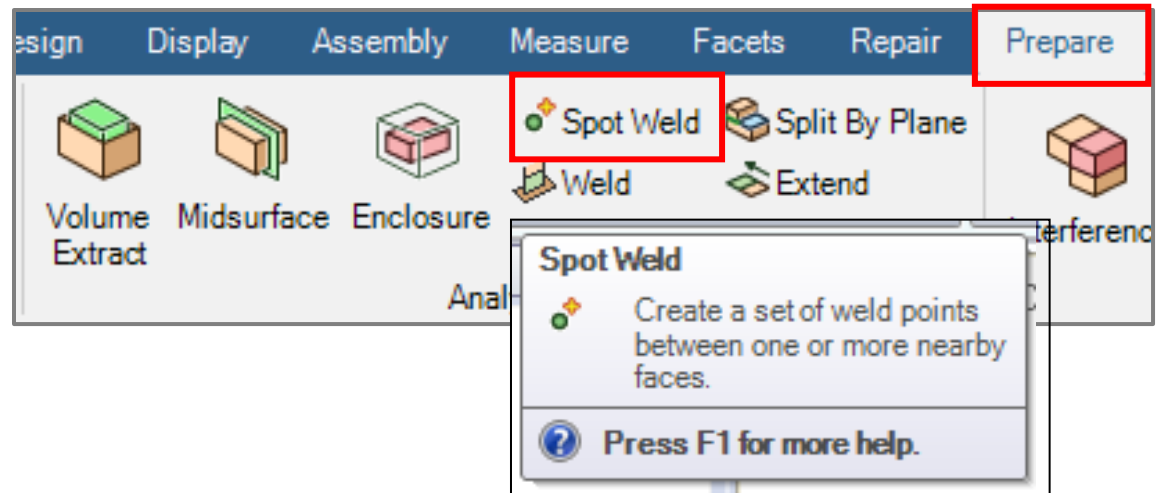
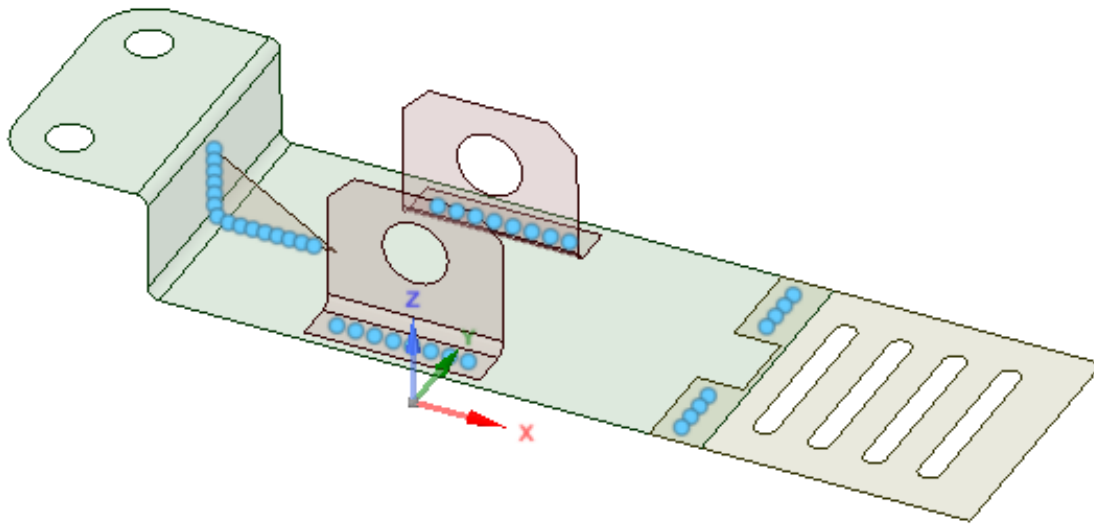
# Overall Process



# / Spot Weld (1)



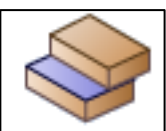

- Spot Weld:

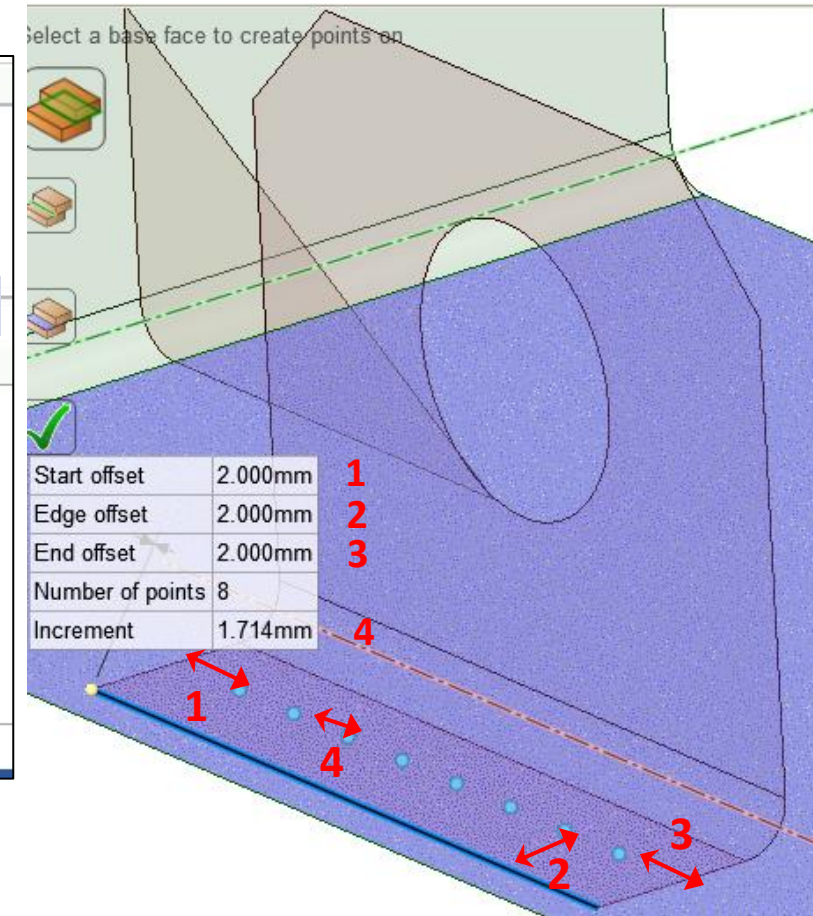
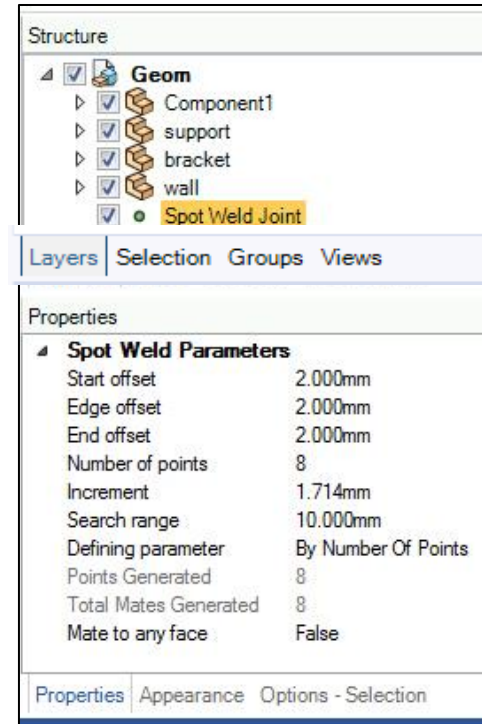
- Spot weld tool is located on **Prepare** ribbon
- Spot welds are applied to the edges of a face of **Surface Bodies**. These will represent spots that share a common node between the two parts
- The property will automatically pass over as an additional connection between the parts to Workbench



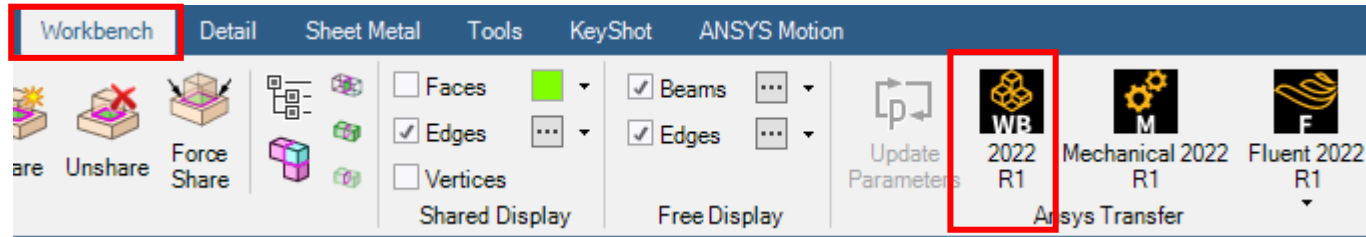
# / Spot Weld (2)

- Spot Weld Tool Guide and options

	Use the <b>Select Base Faces</b> tool guide to select the face or faces on which the weld points will be defined. You should select a single face or a chain of tangent faces.
	Use the <b>Select Guiding Edges</b> tool guide to define the edge along which the weld points will be defined.
	Use the <b>Select Mating Faces</b> tool guide to change the mating face from the face that is automatically detected. You can select more than one face. Clicking on a mating face removes all previously selected faces and holding Ctrl adds a face.
	The <b>Complete</b> tool guide completes the spot weld definition.

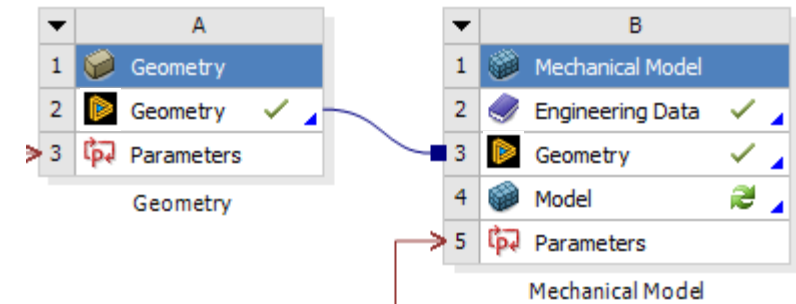


# Spot Weld (3)



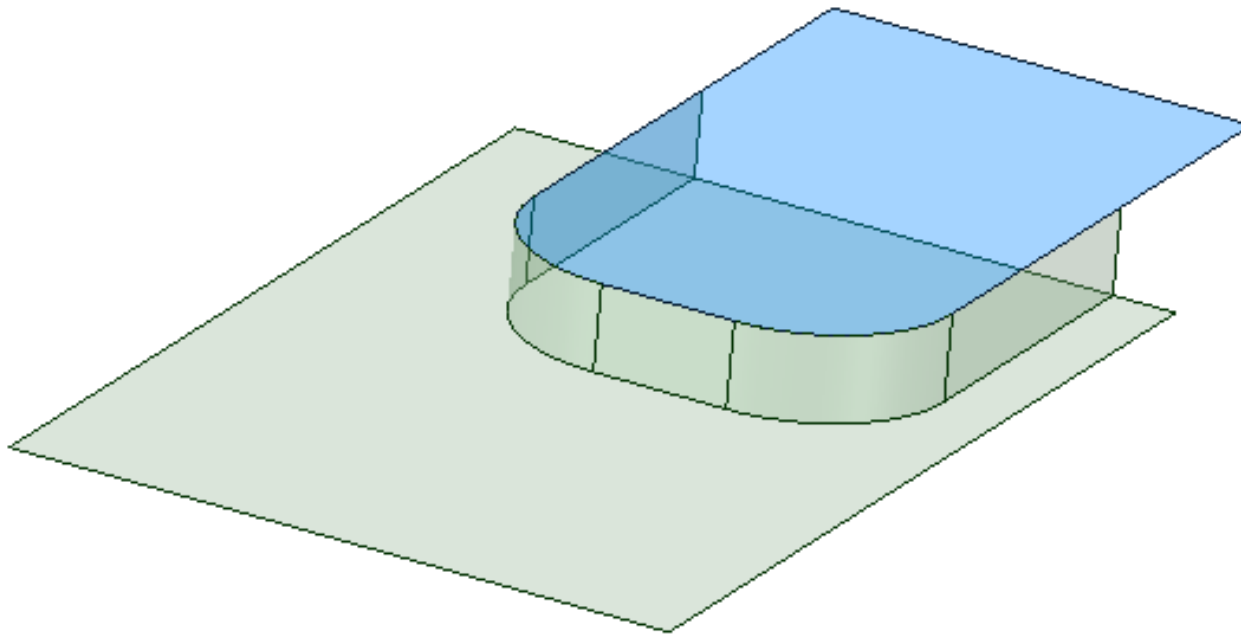
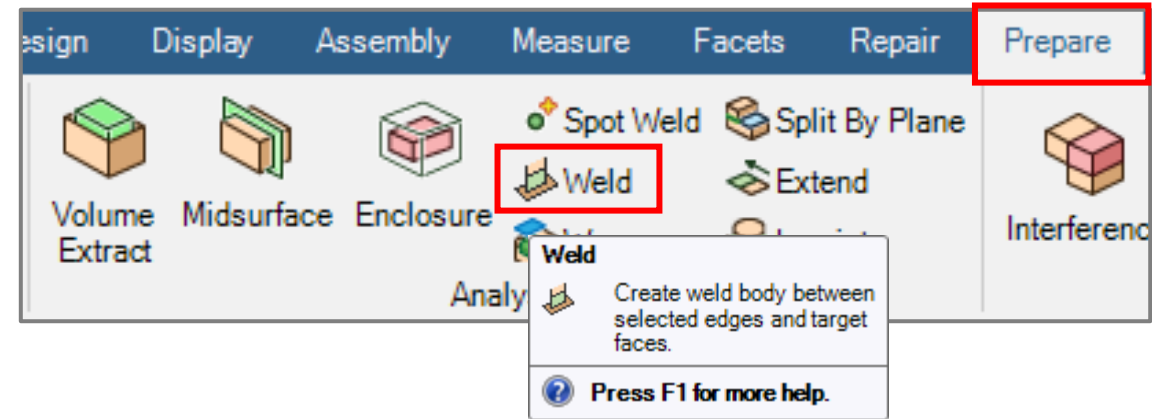
- Transfer to Workbench

- Expanding the structure tree shows all of the spot welds under the connections. Each pair of points will create its own spot weld.
- **Note:** These are some limitations when sending spot welds to Workbench
  - Only points with mates can be used for simulation
  - You may place weld points between multi-body parts if the two bodies belong to different parts. Spot welds defined between bodies in the same part are not transferred to simulation.
  - You can approximate seam welds by placing weld points on the guiding edge with an offset of zero, if no mating face is found on either side of the base face.








# / Weld (1)

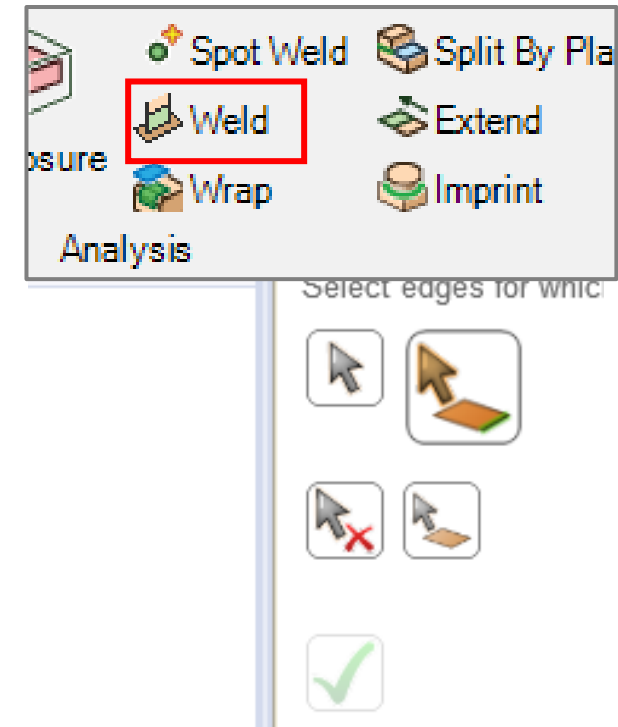
- Surface bodies can be weld to other surface bodies or to solid faces.
- Surface bodies can be weld manually or automatically



# / Weld (2)

- Weld Tool Guide and options

	The <b>Select Problem</b> tool guide is used for selecting sites found and creating welds at the sites
	The <b>Select Edges</b> tool guide is used for manually selecting edges where welds will be created
	The <b>Select Target Faces</b> tool guide is used for selecting the faces where welds will attach
	The <b>Exclude Problem</b> tool guide is used for excluding sites found
	The Complete tool guide creates the weld

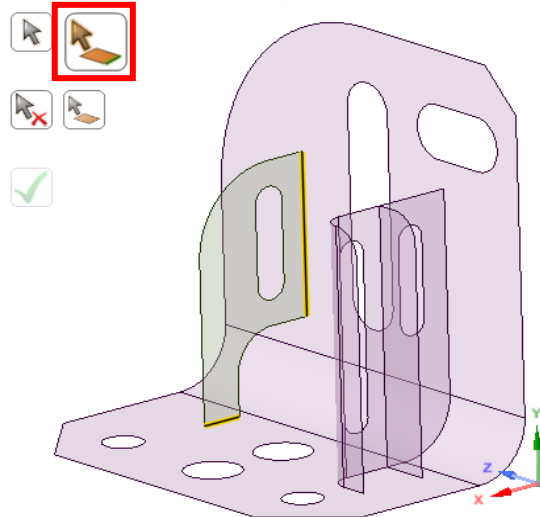




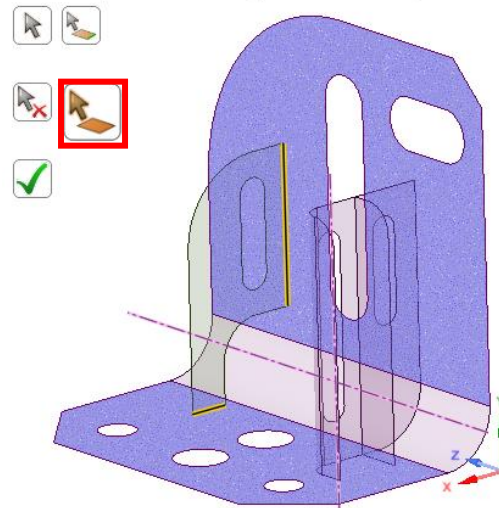
# / Weld (3)

- To weld surface bodies manually
  - Click on the Weld tool in Prepare tab
  - Select edges where the weld body will be created, ctrl-select or loops for multiple selection
  - Click the 'Select Target faces' tool guide
  - Select the face or faces where the Weld body will attach
  - Click on complete tool guide to create the Weld body

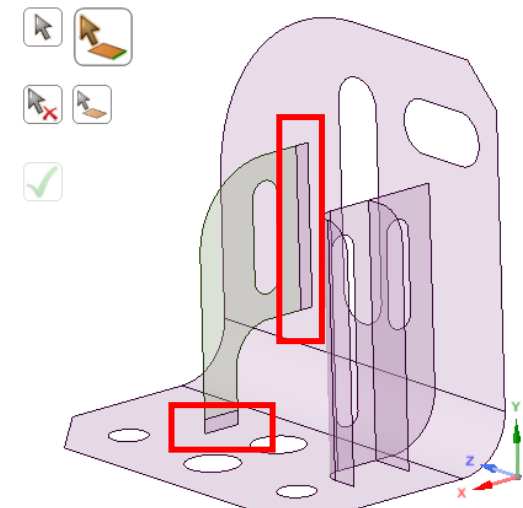
Select edges for which the weld body is to be created.



Click on Complete to create weld(s) from the selected edges and faces.

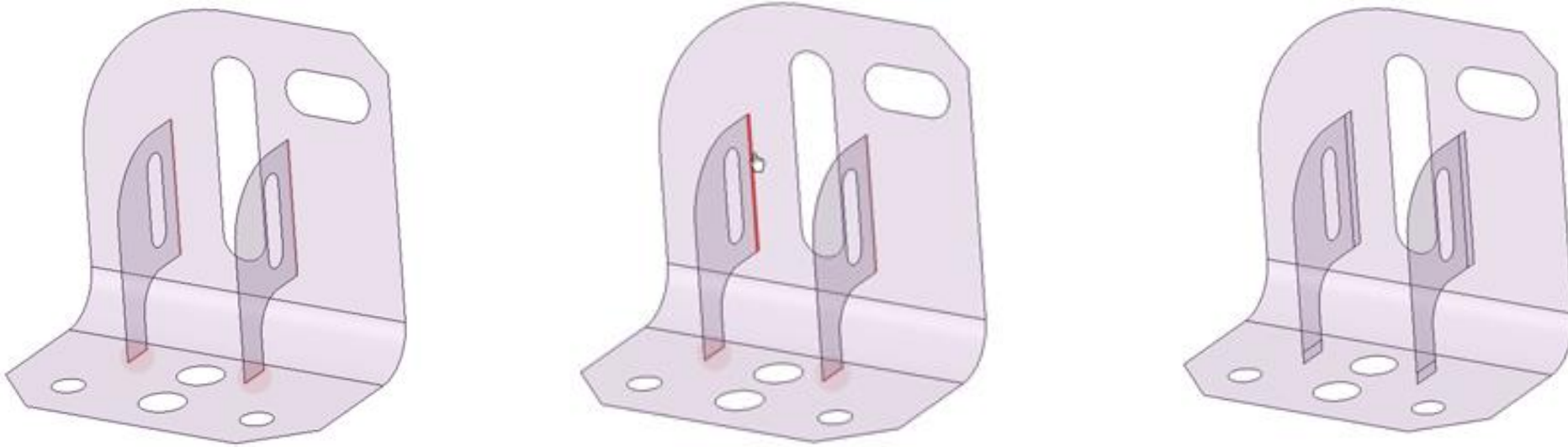


Select edges for which the weld body is to be created.



## / Weld (4)

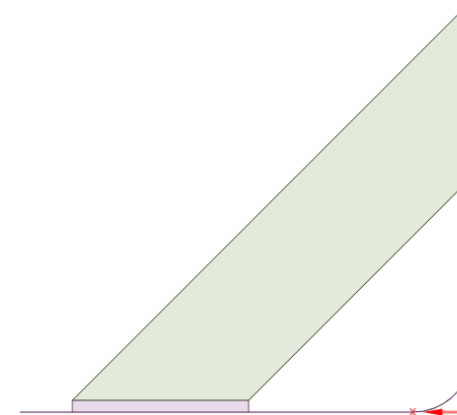
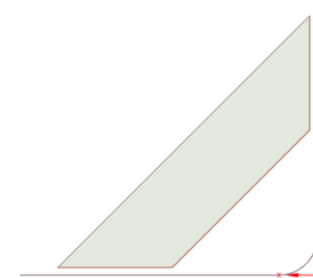
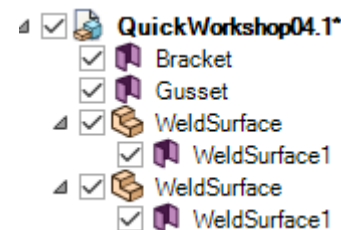
- To find potential weld sites
  - Click on the Weld tool in Prepare tab
  - The Select Problem tool guide is active by default
  - Based on the Maximum Length, potential weld sites are highlighted
  - The Maximum Weld Length can be changed if needed
  - Select individual sites for weld creation or click Complete to create welds on all the sites found



# Quick Workshop 04.1

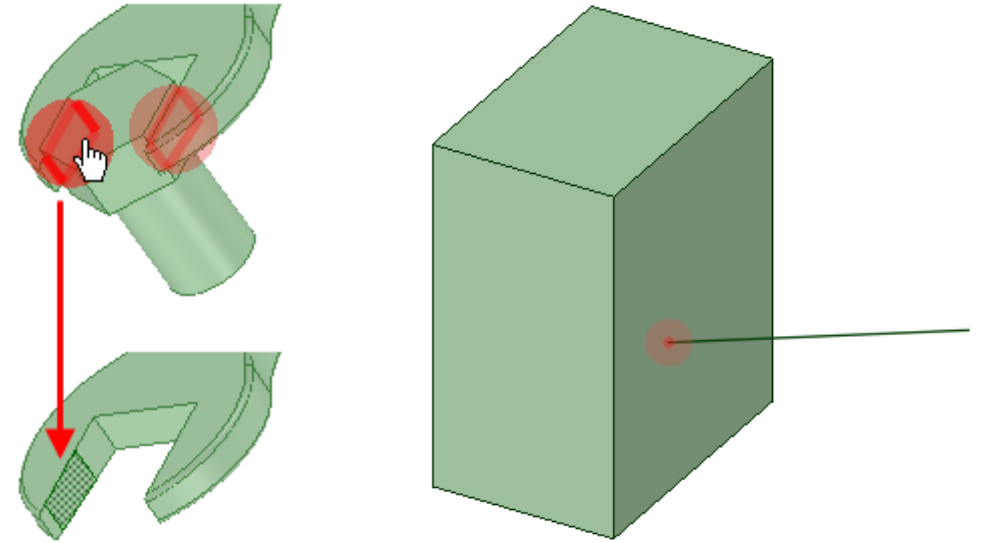
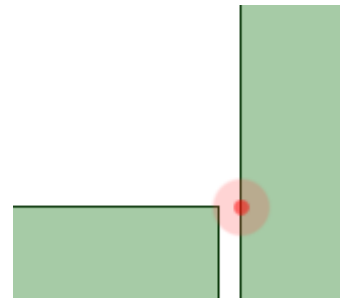
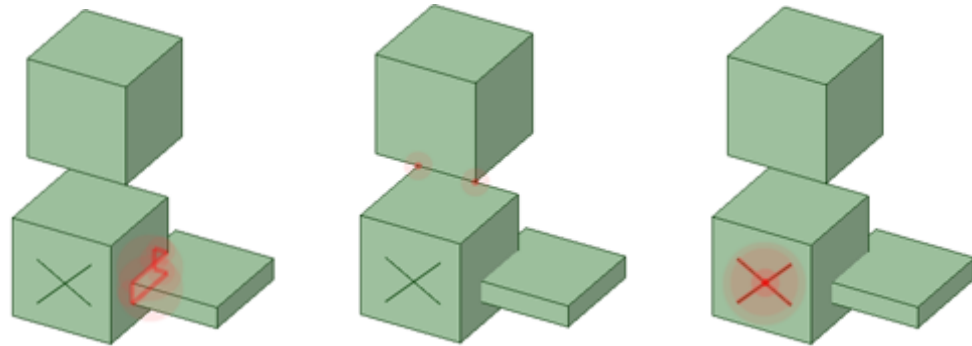
- Open a new SpaceClaim project and import the geometry QuickWorkshop04.1
- Click on the 'weld' function in the Prepare tab
- With the Maximum Length set to 10 mm, 3 edges are selected
- Change the Maximum Length to 5 mm
- 2 edges are selected now
- Click on the green tick mark to create the 2 weld faces

Find Options  
Maximum Length



# / Imprint

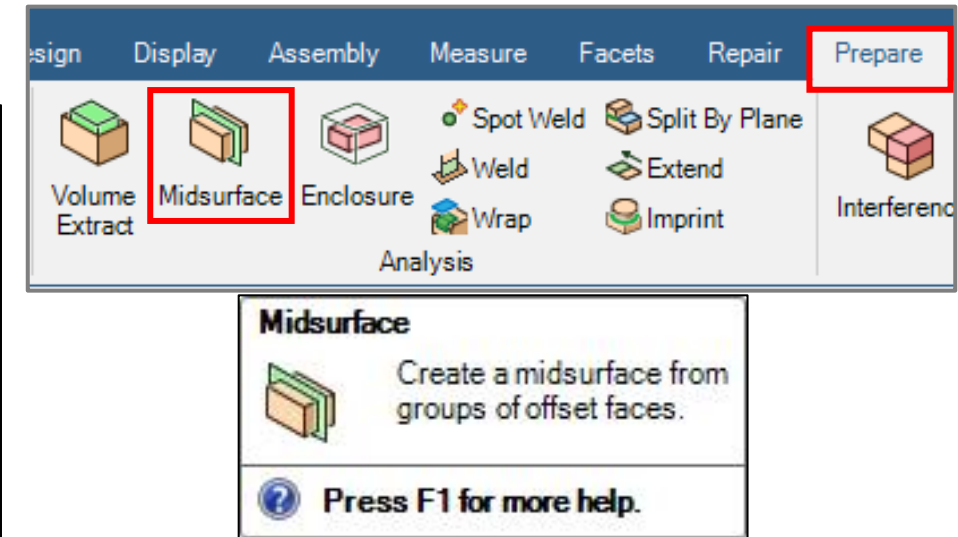
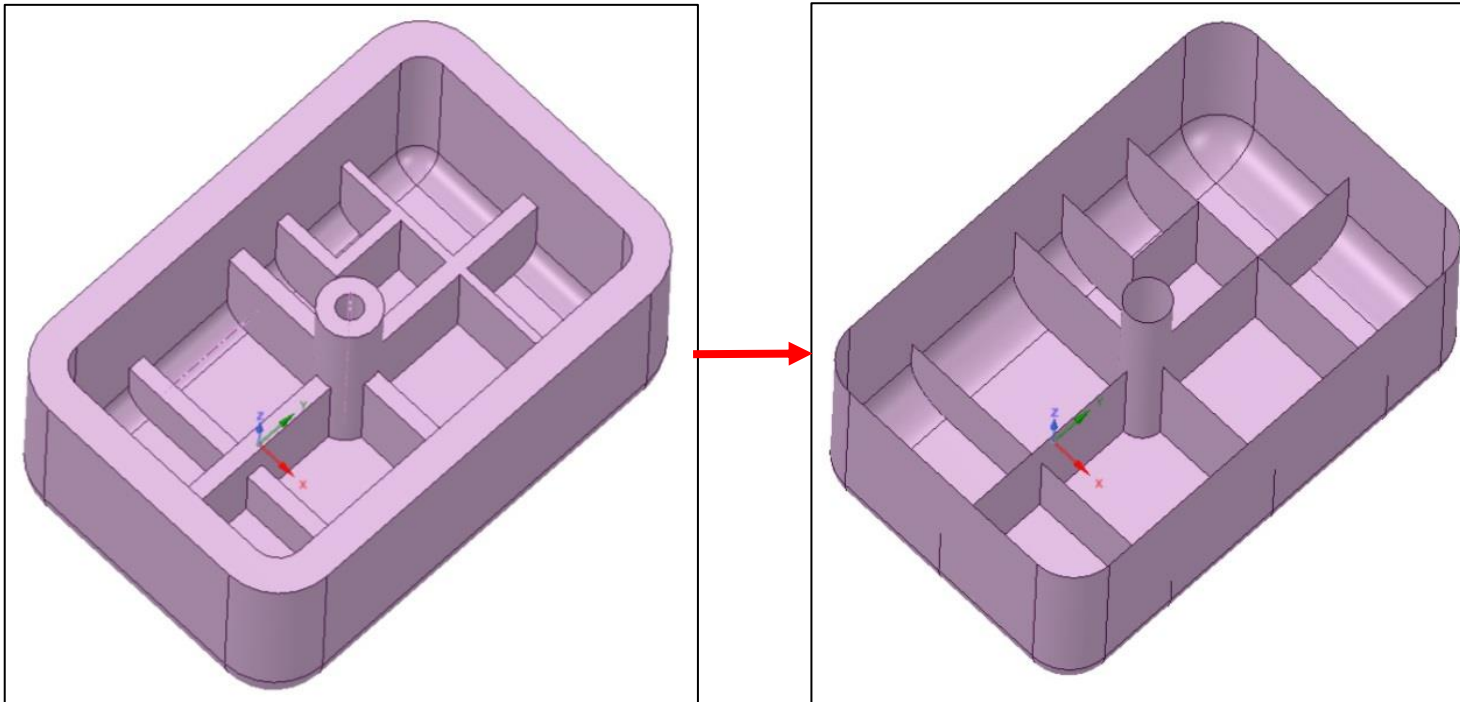
- The **Prepare / Imprint** tool detects coincident faces, edges, and vertices between bodies and imprints them onto the coincident face within the given tolerance
- Options:
  - Face: finds intersections between two faces
  - Edge: finds intersections between two edges
  - Beam/Curve: for intersection between curves/beams



# / Midsurface (1)


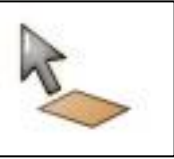


- Midsurface

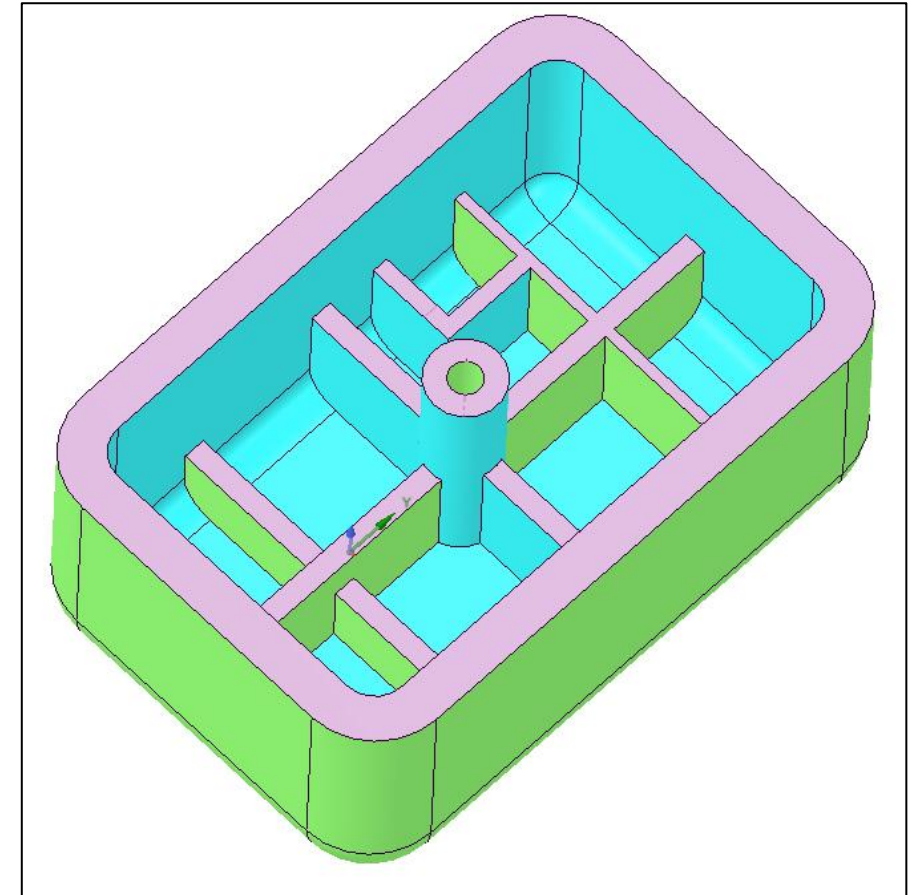
- **Midsurface** tool is used to simplify the model and mesh with shell element in the simulation
- Midsurfaces can be created by either selecting faces, or having SpaceClaim find thicknesses within a given range



# / Midsurface (2)

- Midsurface Tool Guide

	The <b>Select Faces</b> tool guide is active by default. This tool guide allows you to select a pair of offset faces, and all other face pairs with the same offset distance are automatically detected.
	The <b>Add/Remove Faces</b> tool guide allows you to select additional faces to offset or remove detected face pairs from the selection.
	The <b>Swap Sides</b> tool guide allows you to switch the face pairs. You may need to do this when you detect pairs with more than one offset distance, and the offset relationships are incorrectly detected.
	The <b>Complete</b> tool guide creates the midsurface faces.




**Note:** Selected face pairs are highlighted in blue and green color where blue color indicates the positive normal side of the midsurface

# / Midsurface (3)

## • Midsurface Options

<b>Use selected faces</b>	Select this option to create midsurfaces for only the faces you select.
<b>Use range</b>	Select this option to create midsurfaces on all faces in the specified thickness range.
<b>Thickness tolerance</b>	Change the value of this option to detect offset spline faces with an offset value within the tolerance amount.
<b>Create midsurfaces in</b>	Select <b>Same component</b> to create the midsurfaces in the same component as the part you selected for midsurfacing. Select <b>Active component</b> to create the midsurfaces in the active component.
<b>Create midsurfaces at</b>	Bottom, Middle or Top. Automatically assigns property to midsurface body during extraction
<b>Group midsurfaces</b>	Select this option to create midsurfaces in a new sub-component. Deselect the option to create the midsurface objects in the component you select in the option above (same component or active component).
<b>Extend Surfaces</b>	Automatically extend surfaces (On by default)
<b>Trim Surfaces</b>	Automatically trim the midsurface to the extent of the original body

Options - Midsurface 

Select Options

☒ Use selected faces

☐ Use range

Minimum thickness:

Maximum thickness:

Thickness tolerance:

Create Options

Create midsurfaces in:

☐ Same component

☒ Active component

Create midsurfaces at:

☐ Bottom

☒ Middle

☐ Top

☐ Group midsurfaces

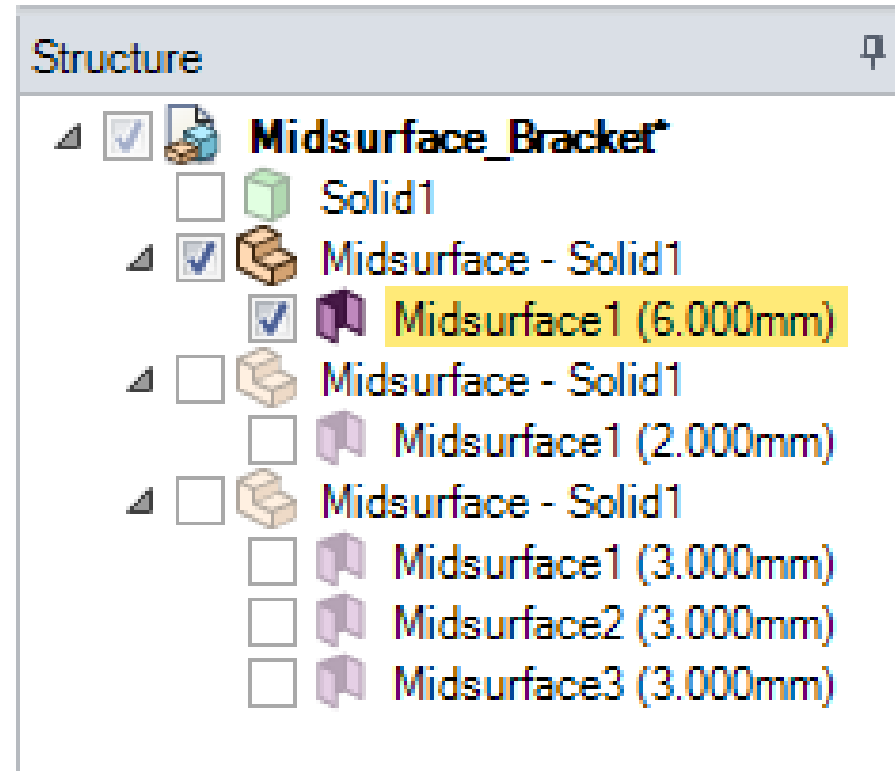
☒ Extend surfaces

☐ Trim surfaces



# / Midsurface (4)

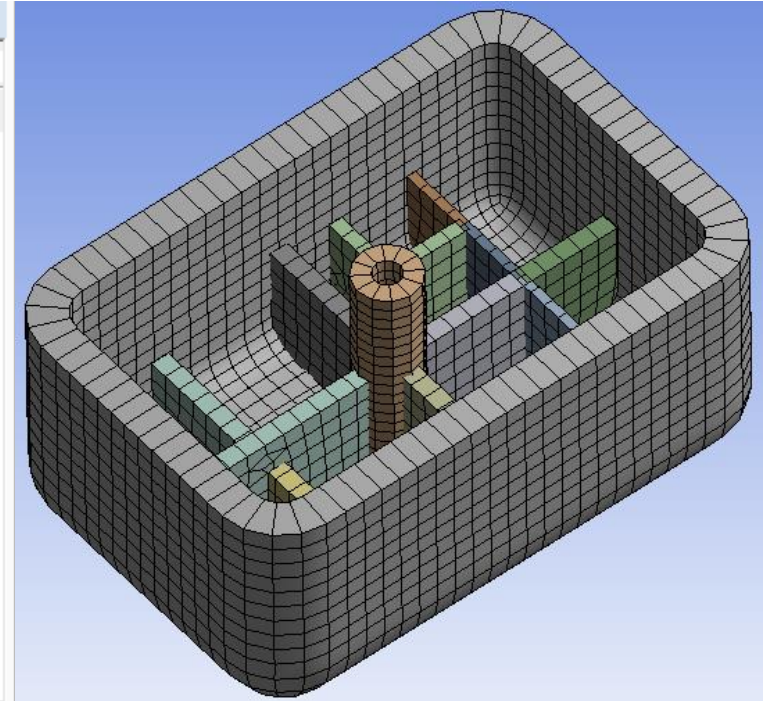
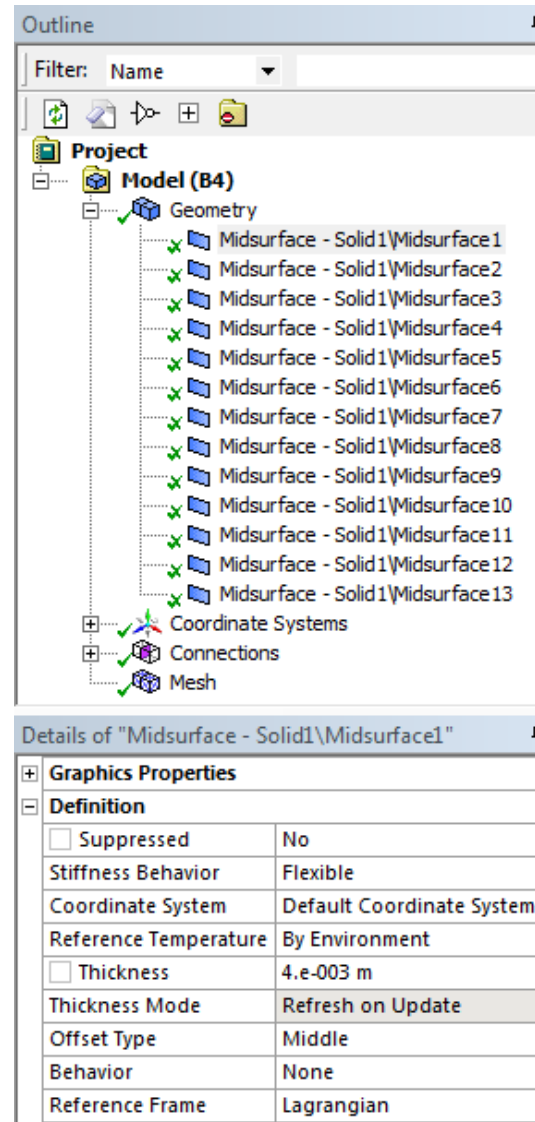
- Midsurface thickness
  - Midsurface thickness is displayed as part of the name in the Structure tree.
  - Body thickness is now stored with overrides for individual faces that have a different thickness (if any)





# / Midsurface (5)

- Transfer to Workbench
  - Only Visible bodies in SpaceClaim are transferred to Workbench.
  - Thickness of the midsurfaces created in SpaceClaim is also automatically transferred to Workbench on import.

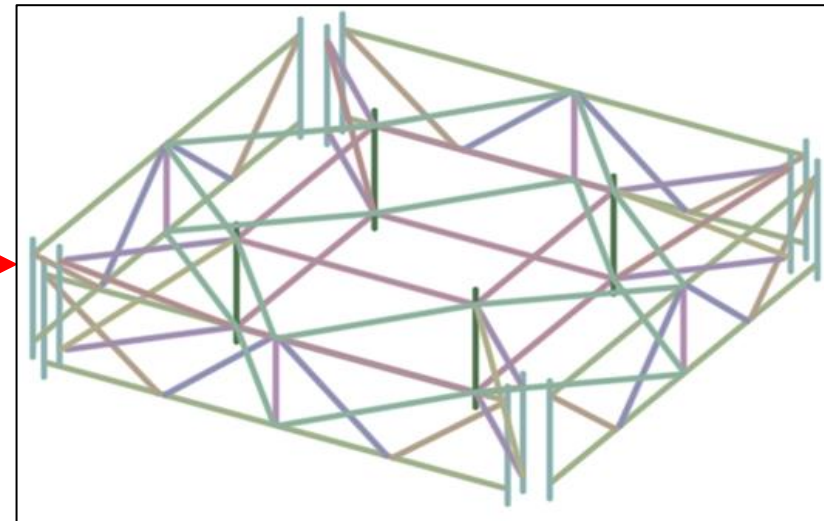
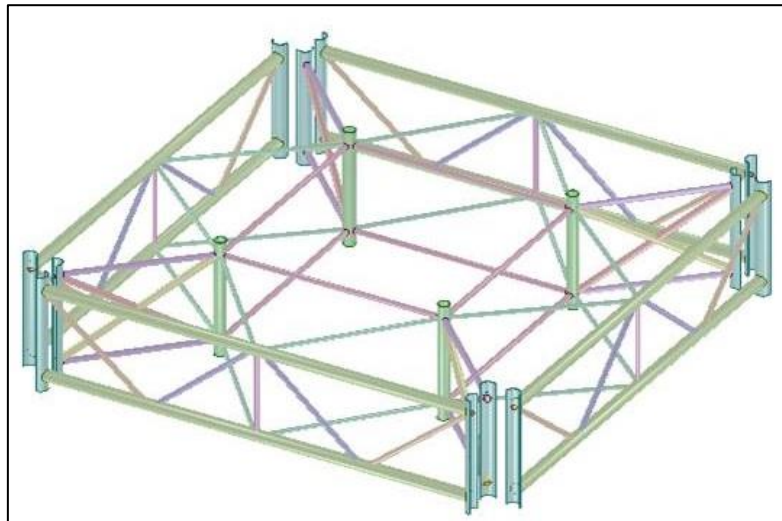
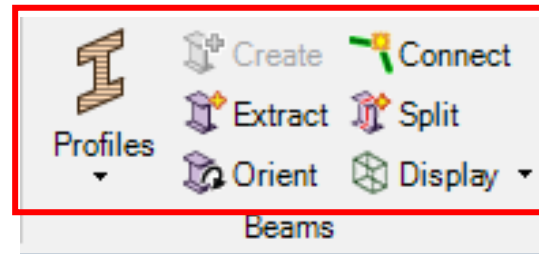


# / Beam Extraction (1)

- Beams Group

- The Beams Group on the Prepare ribbon has the following tools which can be used for creation, extraction and modification of the beams

- Profile
- Create
- Extract
- Orient
- Connect
- Split
- Display



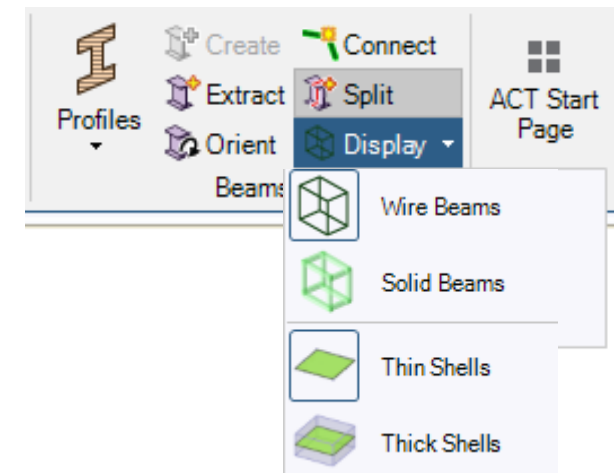
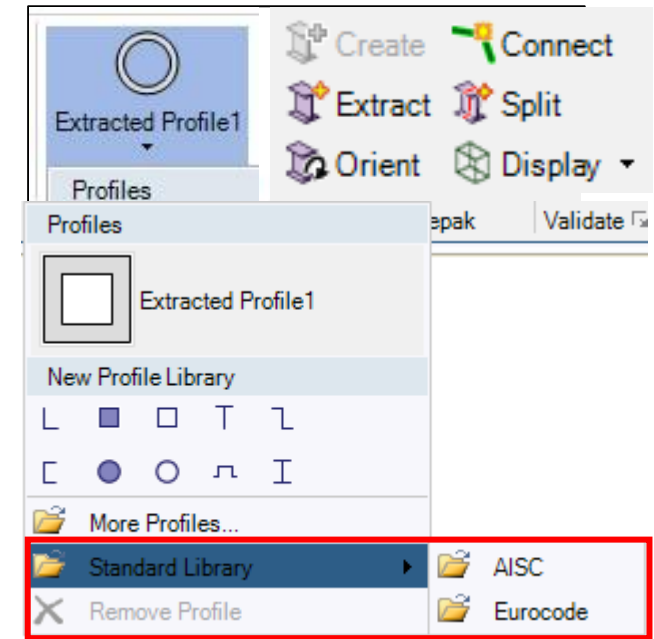
# / Beam Extraction (2)

- Profiles

- This tool helps to define profiles (cross-sections) of beams
- You can select a profile either from a list (which is populated automatically on extraction of beams from solid bodies) or from a library or from an imported set from other \*.scdoc files
- Profiles can be linked to more than one beam in the geometry and hence when the profile is changed for one beam, it is changed for other associated beams as well
- It can also be used to remove a profile from the Beam Profiles folder

- Display

- This tool from the Beam Profiles folder, helps to Toggle the display from Wire Beams to Solid Beams (Lightweight representation of the beam) to help visualize the structure.

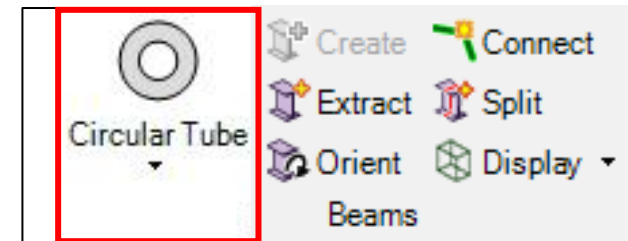



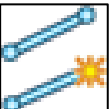
# / Beam Extraction (3)

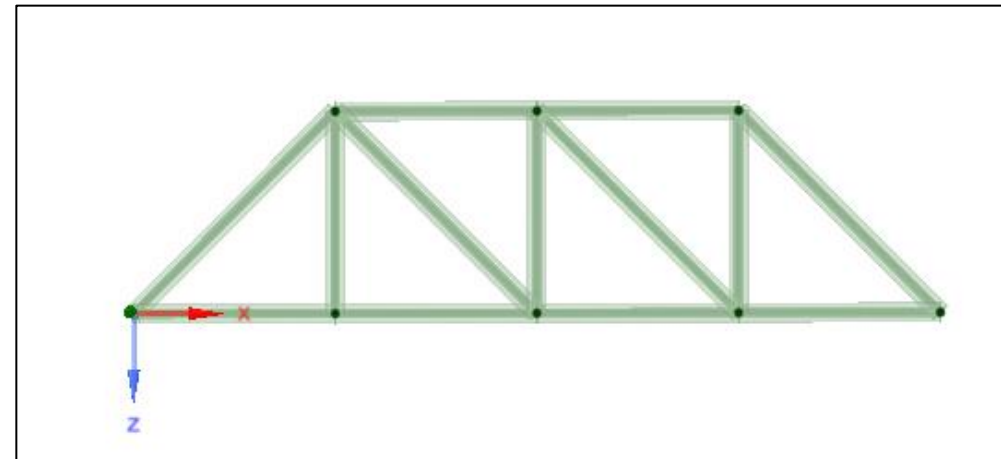
- Create

- The Create tool won't be active until a profile is selected, which adds the profile to the design document.
- After selection of profile, beam path can be defined using edges or points in the document.
- The points of reference can be intersection points or midpoints on edges and other beams

- Tool Guide



	Use the Select <b>Point Chain</b> tool guide to create a beam along an edge or a series of points that you select.
	Use the Select <b>Point Pairs</b> tool guide to create a beam between two points.

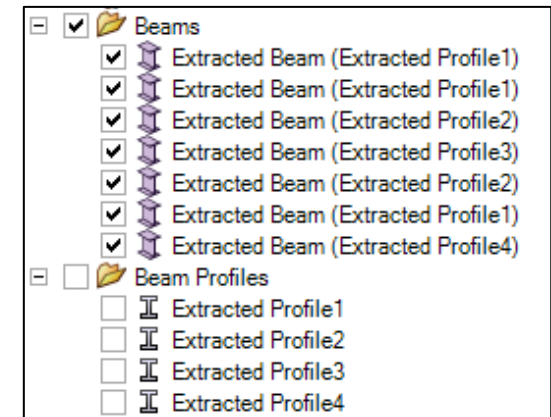
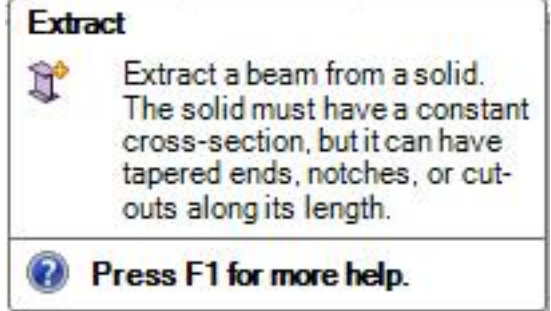
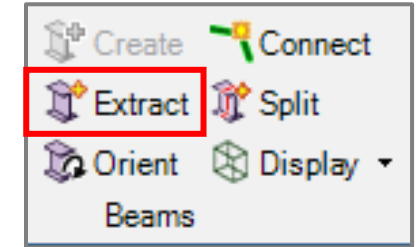
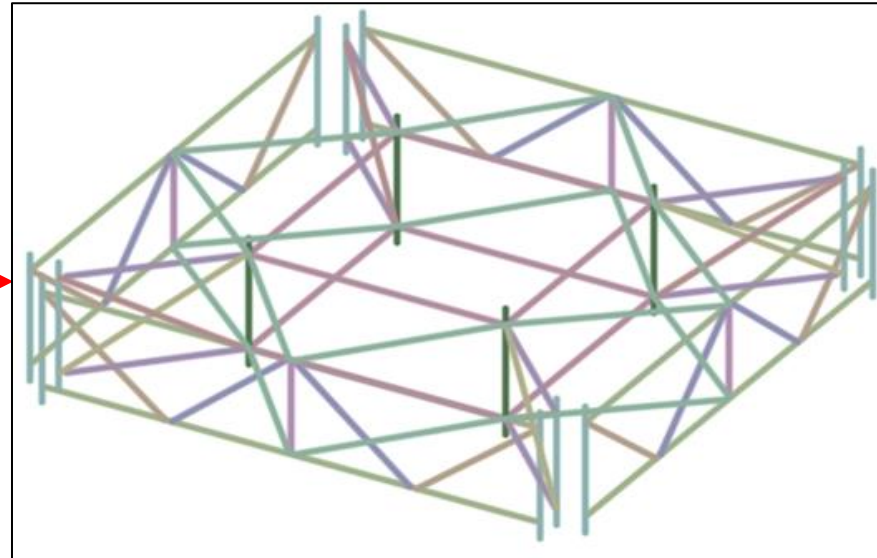
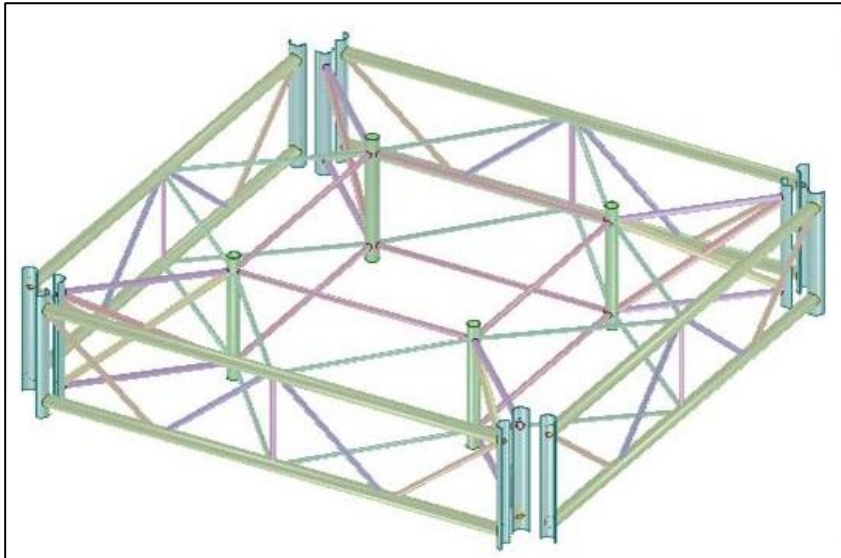




# Beam Extraction (4)

- Extract



- This tool places a 3D line in place of the solid, with cross sectional properties from the original solid

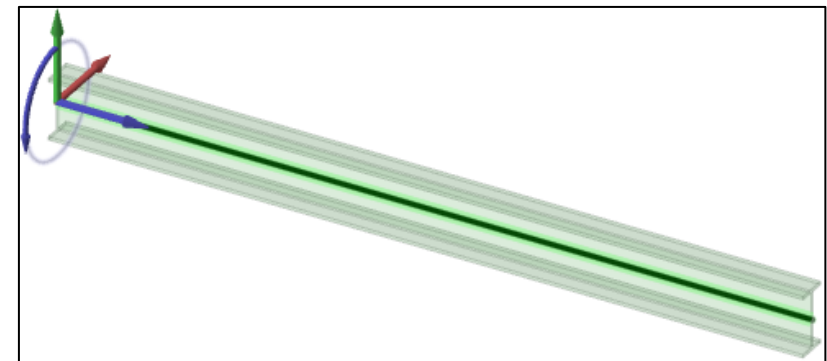
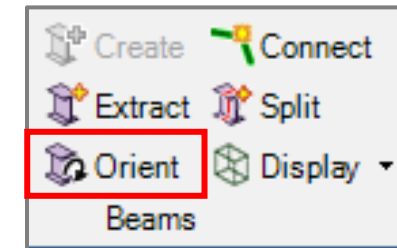


# / Beam Extraction (5)

- Orient

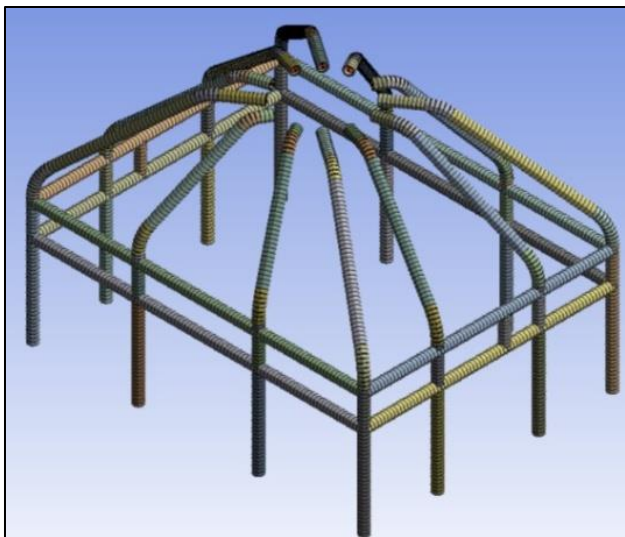
- This tool is used to change the direction of a beam, rotate it around its anchor point, and offset it from its anchor point.
- Orient tool works like move tool for beam profiles
- Tool Guide for Orient tool

	The <b>Select</b> tool guide is active by default. This tool guide allows you to select the beam object that you want to reorient.
	The <b>Orient to Object</b> tool guide allows you to select a face, edge, or axis and orient the beam in that direction.



# Beam Extraction (6)

- Transfer to Workbench
  - Line bodies should be turned ON in SCDM options in order for beams to be imported into Workbench (explained later)
  - Beams can be extracted from existing solids
  - Beams are associated with beam profiles which determine the properties of each beam after being transferred into Workbench



SpaceClaim Options

Workbench options

- ☒ Solid bodies
- ☒ Surface bodies
- ☒ Line bodies
- ☒ Attributes
- ☐ Named selections
- ☐ Material properties

Attribute key: SDFEA;DDM

Named selection key: NS

Parameters: None

Parameter key: ANS;DS

Advanced Geometry Options

Analysis type: 3D

Mixed import resolution: None

☐ Coordinate systems

☐ Work points

Coordinate system key:

Export to Workbench

☒ Merge connected beams

☐ Always use SpaceClaim's reader when possible

File Type: SpaceClaim Reader

Workbench Plug-In/Reader

OK Cancel

Outline

Filter: Name

Project

Model (B4)

Geometry

Geom

Workbench

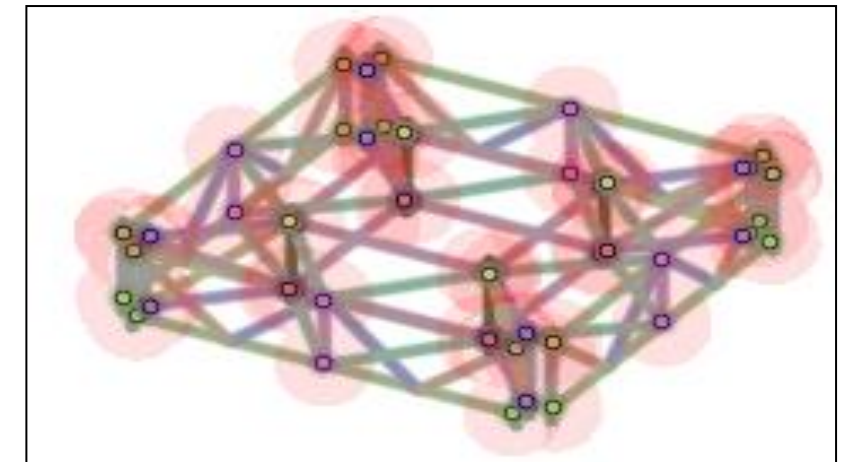
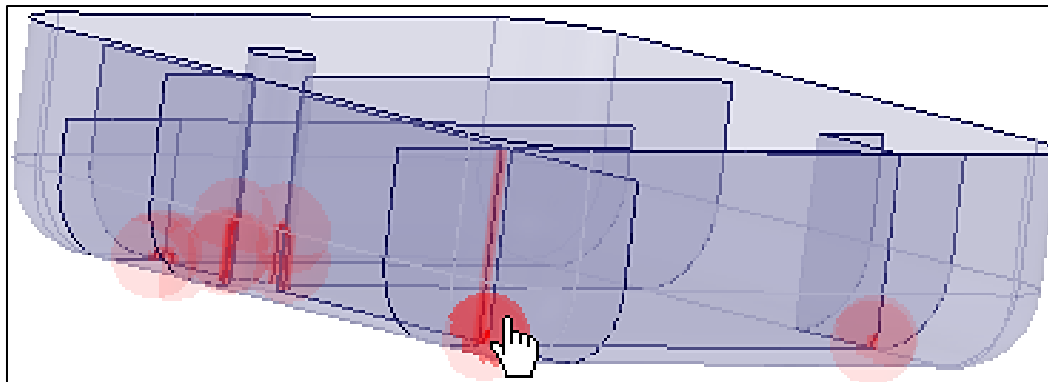
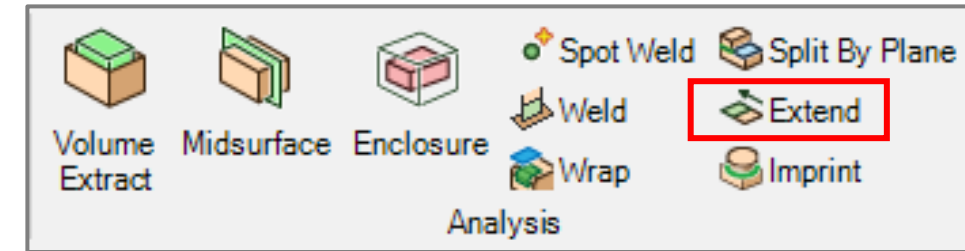
Details of "Extracted Profile1"

Graphics Properties	
Definition	
Material	
Bounding Box	
Properties	
<input type="checkbox"/> Volume	1.4353e+006 mm <sup>3</sup>
<input type="checkbox"/> Mass	11.267 kg
<input type="checkbox"/> Length	336.02 mm
<input type="checkbox"/> Cross Section	Extracted Profile1
<input type="checkbox"/> Cross Section Area	4271.4 mm <sup>2</sup>
<input type="checkbox"/> Cross Section IYY	5.5231e+006 mm <sup>2</sup> .mm <sup>2</sup>
<input type="checkbox"/> Cross Section IZZ	5.5231e+006 mm <sup>2</sup> .mm <sup>2</sup>
Statistics	

# / Extend (1)

- Extend


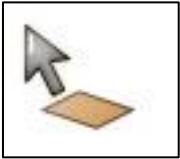


- This tool is helpful to extend or trim surfaces and merge them with nearby parts, or to extend or trim sketch curves and beams
- The tool automatically detects faces/curves that can be extended or trimmed and highlights them
- Also can be used to extend or trim a surface/curve
- This tool only works with surface parts and sketch curves

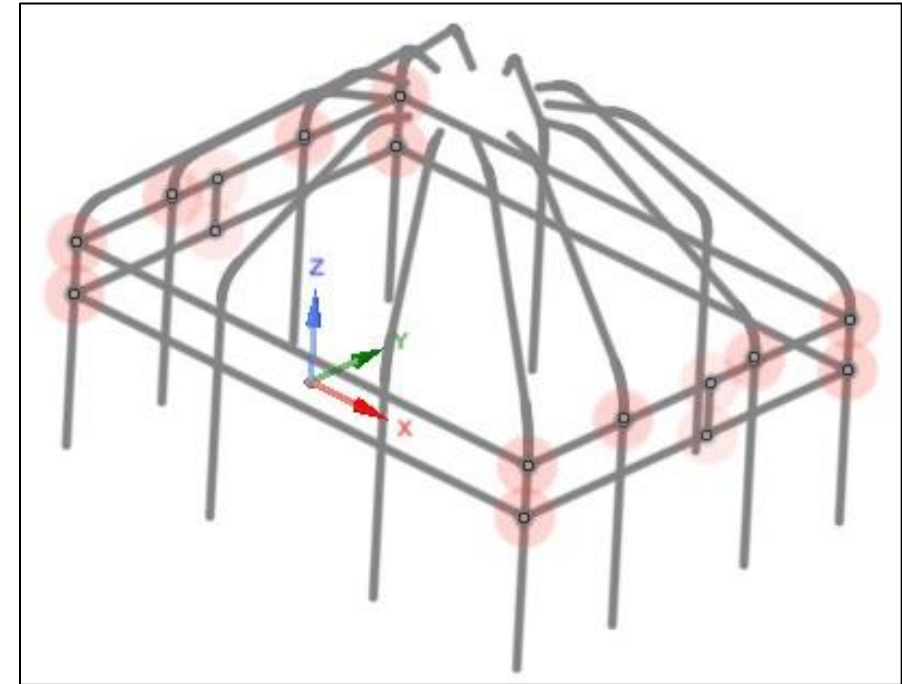




## / Extend (2)

- Tool Guide

	The <b>Select Problem</b> tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The <b>Select Geometry</b> tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
	The <b>Exclude Problem</b> tool guide allows you to select problem areas to be excluded from selection or fixing.
	The <b>Complete</b> tool guide merges or trims the highlighted surfaces.




# / Extend (3)

- Options

<b>Maximum distance</b>	The maximum distance between parts. The tool will search for adjacent faces again if you change this value.
<b>Trim surfaces</b>	Controls whether or not surfaces may be trimmed in addition to extended.
<b>Partial intersections</b>	Controls whether or not faces that partially intersect are detected.
<b>Same body</b>	Allows a surface to be trimmed or extended by a face or edge on the same body.
<b>Extend to curves</b>	Finds surfaces to extend to curves when the curve is in the same plane as the surface.
<b>Merge after extend or trim</b>	Merges bodies, if possible, when you trim or extend an edge on one surface body up to a face or edge on another body.

Options - Find/Fix

 Find Options


Maximum distance:

☐ Trim surfaces

☐ Partial intersections

☐ Same body

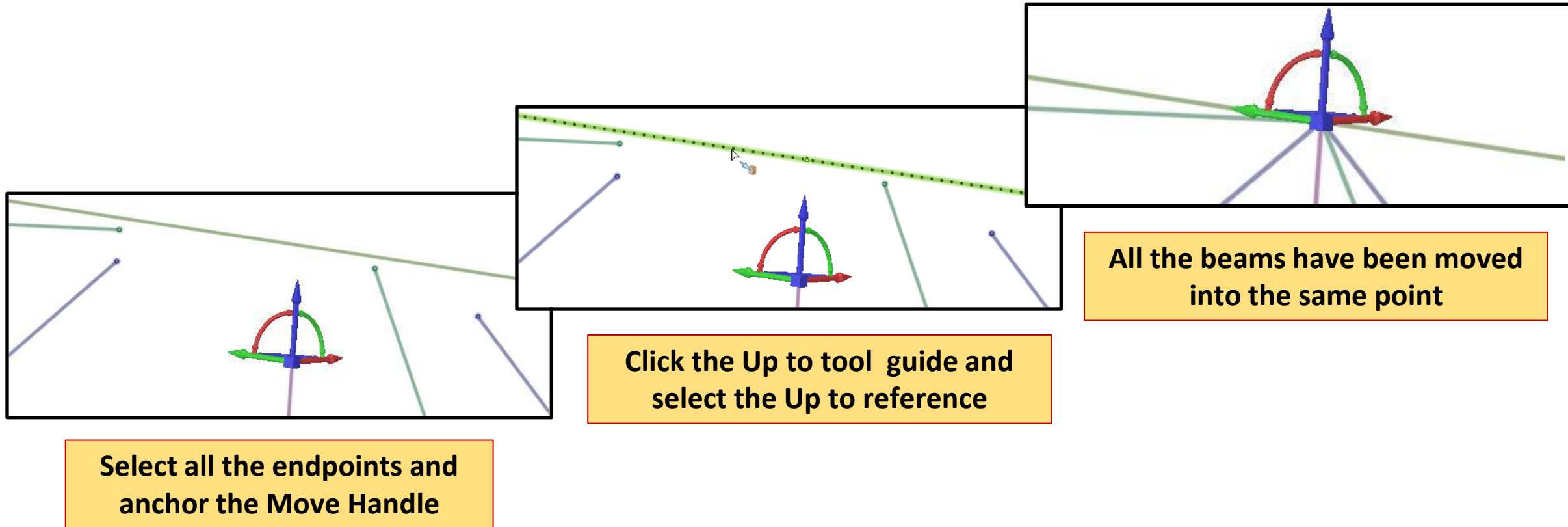
☐ Extend to curves

 Fix Options

☐ Merge after extend or trim

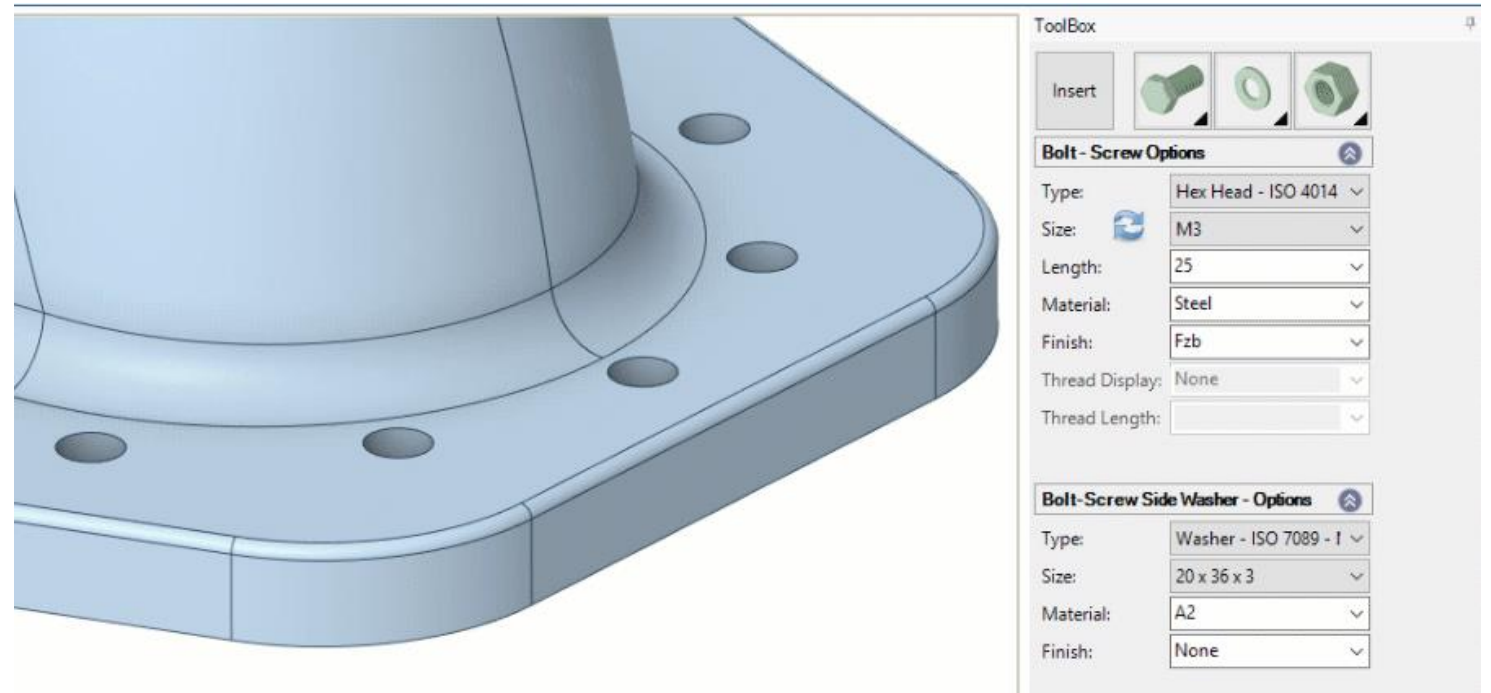
# Modeling Tools – Move

- The process of moving Curve/Beam endpoints to a single point has been enhanced



- This app can be added to SpaceClaim with the following key features:
  - “Fasteners” tool
    - Add hardware, such as bolts, nuts, and washers into your assembly
    - Automatically add hardware to all holes on a given face

## SC Toolbox for SpaceClaim V1

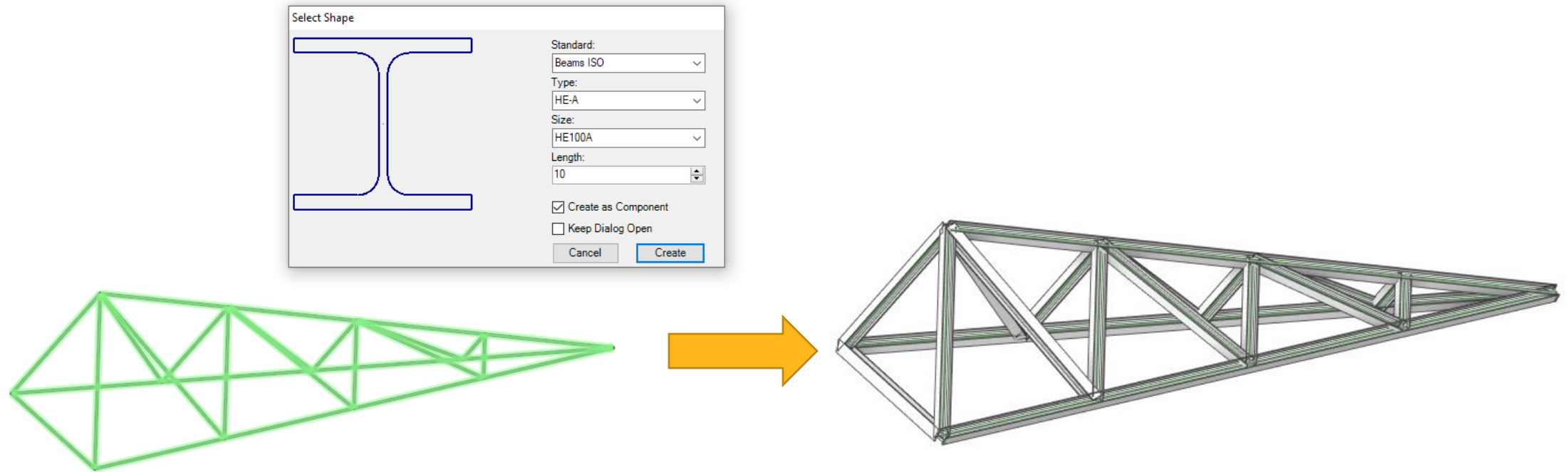


# SpaceClaim Toolbox in App Store



- “Shapes” tool
  - Insert solid beam profiles into your assembly or automatically align them to existing curves

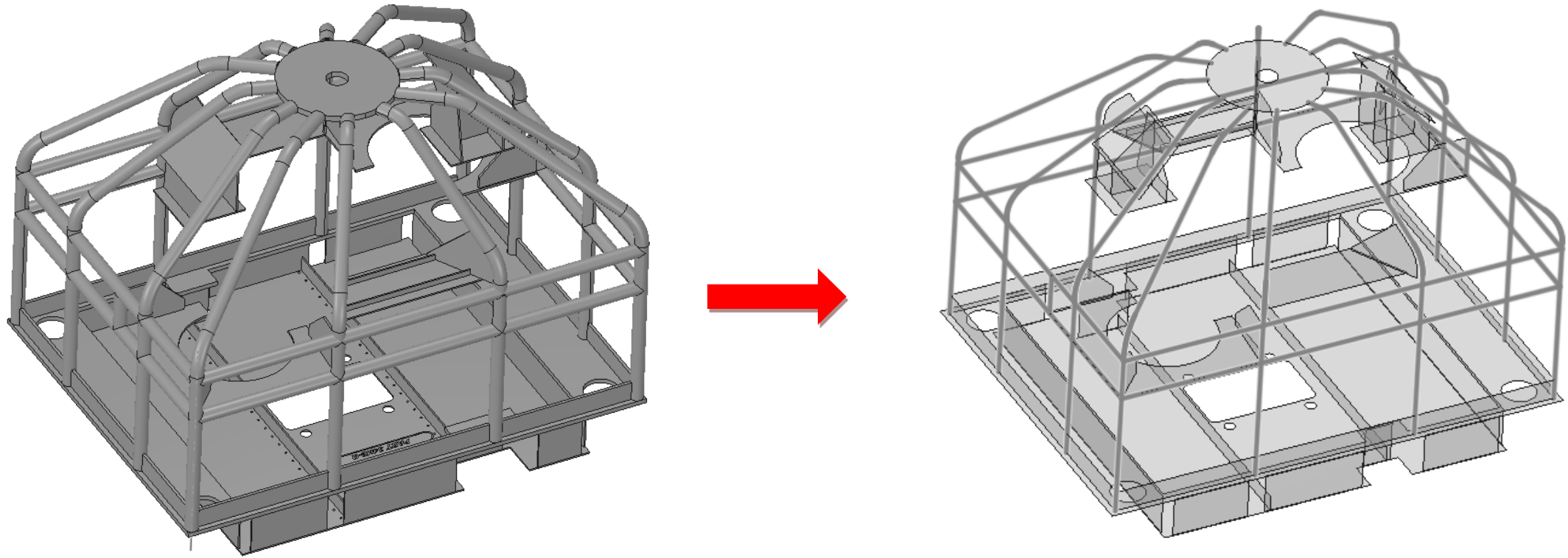
**SC Toolbox for SpaceClaim  
V1**



# / Summary

- The weld feature will create a weld body between selected edges and target faces
- Midsurface will create a midsurface from a group of offset faces
- Beam Extract will extract a line beam from a solid body with its profile and will allow of editing the profile
- Extend will extend surface edges or sketch curves up to intersecting bodies

## Workshop 4.1 Preparing for FEA Analysis





**End of presentation**